PROTEL-AUTOTRAX

PROTEL TECHNOLOGY

PROTEL-AUTOTRAX

Reference Manual

A Printed Circuit Board Design Program

Copyrights

Software Copyright © 1988 Protel Technology Pty Ltd

Original Reference Manual Copyright © 1988 Protel Technology Pty Ltd

PROTEL TECHNOLOGY PTY LTD
Technopark, Dowsings Point,
Hobart, Tasmania, Australia.

Tel. National (002) 73 0100
Tel. International + 61 02 73 0100
Telex AA58260 CRTEC
Facsimile National (002) 73 1871
Facsimile International + 61 02 73 1871

PROTEL TECHNOLOGY INC.
50 Airport Parkway,
San Jose, CA. 95110.

Tel. 408 43 77771
Telex 510 100 7634
Facsimile 408 43 77777

Members of the CRITEC CORPORATION

Epson FX-80® is a registered trademark of Espon America, Inc.

IBM® is a registered trademark of International Business Machines Corporation.

PC-DOS® is a registered trademark of International Business Machines Corporation.

Microsoft® is a registered trademark of Microsoft Corporation.

MS-DOS® is a registered trademark of Microsoft Corporation.

HP® is a registered trademark of Hewlett-Packard Corp.

Hercules™ is a trademark of Hercules Computer Technology.

Introduction

Congratulations on your purchase of ***PROTEL-AUTOTRAX***. As you will soon see it contains exciting features which take the hardship out of the process of designing printed circuit boards.

The program has been written by PROTEL TECHNOLOGY PTY LTD an Australian company which specialises in the development of software products for the design of printed circuit boards.

PROTEL-AUTOTRAX is the latest addition to the PROTEL family of CAD software and incorporates a host of ideas which have come from many thousands of international users of the ***PROTEL-SCHEMATIC*** and ***PROTEL-PCB*** programs.

PROTEL-SCHEMATIC is a program which will help you produce excellent quality schematic drawings whether they be for design verification or for a permanent record of a circuit. When used in conjunction with ***PROTEL-AUTOTRAX*** you have a very powerful end-to-end design system. ***PROTEL-SCHEMATIC*** allows you to produce a **netlist** (summary of electrical connections) of the schematic. With its netlist capture feature, plus the automatic placement of components and autorouting of tracks on the PCB, ***PROTEL-AUTOTRAX*** provides all the features you need to produce high quality camera-ready artwork on either an ink plotter, photoplotter, dot matrix or laser printer.

PROTEL-AUTOTRAX now becomes PROTEL's high-end PCB design program whereas ***PROTEL-PCB*** caters for low-end users who just want to produce a quick layout of a PCB without using the sophisticated features of ***PROTEL-AUTOTRAX***.

Before reading further we ask you to read the **Licence Agreement** which is in the front section of this manual. This request is made for your benefit. Information about upgrades to the software and new product information can only be sent to registered users. After reading the licence details please fill in the registration card with your postal details and send it back to us.

TABLE OF CONTENTS

1 Setting up *PROTEL-AUTOTRAX* **13**
Hardware check list 13
Software requirements 14
PROTEL-AUTOTRAX check list 14
Installing the software on a hard disk 15
Starting up *PROTEL-AUTOTRAX* 17

Tutorial section

2 Tutorial: Getting started **21**
Using the keyboard 21
Loading the Demo PCB 23
Using the menus 24
Some special keys 31
Speeding up menu selections 32
3 Tutorial: Laying out a PCB **33**
Getting started 33
Placing components 34
Laying tracks 37
Using the router 39
Connecting to power and ground planes 41

4 Tutorial: Creating new components 43
Getting started 43
Placing some tracks 45
Placing pads and using ***Repeat*** 46
Placing an arc 47
Adding the component to the library 48
5 Tutorial: Using the line editor 49
The input window explained 50
Editing a line 50
Line editor functions summary 52
6 Tutorial: Assigning key macros 53
Defining a macro 53
Using the record facility 55
Defining nested macros 56
7 Tutorial: Using the *TRAXPLOT* program 57
Starting the program 57
A draft print 59
Plotting a board 59

Reference section

8 Manipulating blocks 63
Defining a block 63
'Hiding' a block 64
Moving a block 64
Copying a block 67
Deleting a block 67

Reading a block 68
Saving part of a layout 68
9 Viewing and changing the current settings 69
Setting the cursor mode 69
Setting a floating origin 69
The currently selected layer 70
The current pad type 70
Pad orientation 71
The default track width 71
Checking the text height 72
Checking the width of string lines 72
The default via size 72
The default via hole size 72
10 Deleting items on the board 73
Deleting an arc 73
Deleting a component 74
Deleting a fill 74
Deleting a highlight 75
Deleting a pad 75
Deleting a string 75
Deleting a track 76
Deleting a via 76
11 Changing part of the PCB 77
Changing a component 77
Changing a pad 79
Changing a track 80
Changing a string 80
Changing a via 81

12 Managing files 83
Clearing the memory 83
Issuing DOS commands 84
Checking files stored on the disk 84
Loading a file 85
Specifying a path name for files 86
Leaving the *PROTEL-AUTOTRAX* program 87
Saving your files 87
13 Setting grid sizes 89
Changing the 'snap' grid 89
Changing the visible grid 90
14 Highlighting nets 91
Highlighting a connection 91
Duplicating nets 92
Highlighting a net 93
Creating a netlist 93
15 Getting information about the board 95
Determining the board's dimensions 95
Listing the components already placed 96
Checking highlighted pins 99
Checking the names of library components 96
Listing all the nets 97
Specifying power and ground plane nets 97
Essential information about the board 97
16 Using the *Jump* option 99
Jumping to a component 99
Jumping to a location 100
Jumping to a net 100
Jumping to the origin 101

Jumping to a pad 101
Jumping to a string 102
17 Managing libraries 103
Adding components to the library 103
Browsing through library components 103
Compacting the library 105
Deleting a component from the library 105
'Exploding' a component 106
Choosing a different library file 106
Listing library components 107
Merging libraries 107
Creating a new library 107
Renaming a component 108
18 Moving items on the board 109
Moving an arc 109
Breaking a track 110
Moving a component 110
Dragging the end of a track 112
Moving a fill 112
Moving a pad 112
Re-routing a track 113
Moving a string 113
Moving a track 113
Moving a via 114
19 Managing netlists 115
Placing components automatically 116
Clearing a net 117
Checking the design rules 117
Loading a netlist 117

Hiding nets 118
Identifying nets 119
Determining the connection length 119
Listing net names 119
Optimizing nets 119
Routing a board 120
Showing highlights 120
Setting up the net files 120

20 Routing a board 121

Layer setup 121
Setting up the router passes 122
Separation setup 124
Setting the router variables 125
Routing the board 125
Routing a single connection 128
Routing a connection manually 129
Routing a net 129
Using the pad to pad router 129

21 Placing items on a board 131

Placing an arc 131
Placing a component 132
Creating an external plane 133
Placing a fill 134
Placing a pad 135
Placing a string 135
Placing tracks 135
Placing a via 136

22 Repeating actions 137

23 Changing the setup options 139
Controlling component text 139
Choosing layer colors 140
Selecting menu colors 141
Assigning functions to keys 141
Setting options 146
Defining pad types 148
Choosing the redraw quality 150
Setting string defaults 150
Turning layers on and off 151
24 Using the un-delete command 153
25 Expanding and contracting the display 155
Redrawing the screen 155
Moving around the board 155
Expanding the display 156
Contracting the display 156
Fitting the board on the screen 156
Specifying a zoom factor 156
Selecting a zoom value 157
Specifying a zoom window 157

Utilities section

26 Using the *TRAXPLOT* program 161
Starting the *TRAXPLOT* program 161
The File menu 162
Displaying information about the board 164
Choosing the plot options 165
The setup option 166

Printing or plotting a PCB 170

27 The *PCB3CON* and *PLIB3CON* programs 171

The *PCB3CON* program 171

The *PLIB3CON* program 172

Appendices

A Features of *PROTEL-AUTOTRAX* 175

Important files 175

Hardware and software checklist 178

Hardware requirements 178

Software requirements 180

PROTEL-AUTOTRAX checklist 180

Parameters 181

B RS-232 Considerations 183

Serial versus parallel interfaces 183

Cable connections 184

Baud rate, data bits, etc. 185

What to do in case of problems 186

C Error messages 189

Non-fatal errors 189

Fatal errors 197

D A note to educators 199

Index

Index 203

1 SETTING UP *PROTEL-AUTOTRAX*

PROTEL-AUTOTRAX is a low-cost, yet powerful, software package for producing printed circuit board (PCB) artwork. It is a fully-featured system which runs on the IBM-PC/XT/AT series and close compatibles.

This chapter details the hardware and software you will need to run the program and provides some advice on installing the software.

Hardware check list

You will need the following:

An IBM microcomputer or compatible

2 floppy disk drives or one floppy drive and one hard drive (a hard disk is recommended)

Graphics card (CGA, EGA, Hercules, VGA or Vega Deluxe)

Colour monitor

One parallel printer port

Microsoft mouse or compatible (recommended)

Draft printer

MS-DOS or PC-DOS (Version 2.0 or greater)

You will need access to:

A pen and ink plotter

A Gerber-compatible photoplotter

An Excellon format N/C drill

For the exact hardware requirements, you should check Appendix A at the back of this manual.

The maximum size of a .PCB file which can be loaded by *PROTEL-AUTOTRAX* in the standard 640K RAM is approximately 180K bytes. Files bigger than this will not load and therefore require the use of expanded memory systems (EMS).

If you decide to purchase extra memory, please note that you require an **Expanded Memory Board** rather than an Extended Memory Board. The board should conform to the widely used LIM standard. We have used the 'Hyperam AT' board by Hypertech when designing the program.

Software requirements

In addition to the *PROTEL-AUTOTRAX* software you will need a copy of either the MS-DOS or PC-DOS operating system. The version number should be 2.0 or greater.

This manual assumes that you are familiar with some DOS operations such as creating directories, changing directories, specifying path names, formatting disks, deleting files and copying files.

The software will provide automatic backups of your files, but it is strongly recommended that a regular backup of PCB files is made onto a different disk.

PROTEL-AUTOTRAX check list

When you receive your *PROTEL-AUTOTRAX* package you should find the following contents:

The *PROTEL-AUTOTRAX* manual.

Four diskettes, labelled Program Disk A, Program Disk B, Library Disk and Plot Disk which contain the *PROTEL-AUTOTRAX* programs and their associated files.

The external protection device, which plugs into one of the parallel printer ports of your microcomputer and allows the *PROTEL-AUTOTRAX* program to work. It also prevents unauthorised use of the software.

The Software License Agreement Card, which you should fill out and return immediately. This is vital if you require further support or information about future updates and revisions.

Note that you can make unlimited backup copies of the disks, but the main program will only work if the external protection device is plugged into one of the parallel ports.

Installing the software on a hard disk

You should immediately make a backup copy of each disk and store the originals in a safe place.

If you are using the programs from a hard disk, make a subdirectory called \Protel and copy all the *PROTEL-AUTOTRAX* files into this subdirectory. Create another directory called \PCB to store your PCB files. It is strongly recommended that you store your files in a different subdirectory to the *PROTEL-AUTOTRAX* system files. This makes it easier for you to create backups of your work.

The following instructions provide one means of setting up your hard disk.

Move to the top directory on disk C:

Type: **MD Protel** and press «ENTER»

Type: **MD PCB** and press «ENTER»

Type: **CD Protel** and press «ENTER»

The prompt should now indicate that you are in the \Protel subdirectory (e.g. C:\Protel>)

Put Program Disk A disk in drive A:

Type: **COPY A:*.*** and press «ENTER»

Put Program Disk B disk in drive A:

Type: **COPY A:*.*** and press «ENTER»

Put the Library Disk in drive A:

Type: **COPY A:*.*** and press «ENTER»

Now put the Plot Disk in drive A:

Type: **COPY A:*.*** and press «ENTER»

All the programs and files should now be installed on your hard disk.

If you want the computer to place you in the PCB subdirectory automatically, then you will need to put an extra command in the AUTOEXEC.BAT file.

The command you need to use is:

CD \PCB

If you do not know how to change the file then seek help - it is not a good idea to change this file if you are not confident about the DOS environment.

The next step is to select a graphics driver program which suits the graphics card you have in your microcomputer.

When the *PROTEL-AUTOTRAX* program is loaded it looks for a file called GRAPH.DRV. This file contains the information required to drive the graphics display. If no GRAPH.DRV file is found, the software assumes that you are using a CGA card. If you are using an EGA, VGA, Hercules or Vega Deluxe card then you must create the appropriate GRAPH.DRV file.

The instruction you should type at the DOS prompt is shown in the table below:

Card type	**Instruction**
EGA	**EGA**
VGA	**VGA**
Vega Deluxe	**VEGA**
Hercules	**HERCULES**

If you later wish to return to using a CGA card then type:

CGA then press «ENTER»

and the GRAPH.DRV file will be deleted.

Note that further graphics drivers may be made available in the future.

Starting up *PROTEL-AUTOTRAX*

If you are using a floppy drive system, make sure that Program Disk is in drive A: and Program Disk B is in drive B:. You should also set the DOS PATH variable to point to drive B:. This is best done in the AUTOEXEC.BAT file.

If you are using a hard drive system, you can run the programs from any directory, but you may want to move into the directory in which you want to store your PCB files just to be on the safe side.

If it is the first time you have run the program, select the appropriate graphics driver as described on the previous page.

You are now ready to run the program. Simply type:

TRAXEDIT and press «ENTER»

and the program will start.

If you are using a floppy drive system you can now take out Program Disk A and replace it with the Library Disk.

You should now attempt some of the tutorials which will help you become familiar with the features of *PROTEL-AUTOTRAX*.

TUTORIAL SECTION

2 TUTORIAL: GETTING STARTED

This tutorial allows you to gain an overview of the features of *PROTEL-AUTOTRAX*. It is suggested that you complete the tutorial before attempting to create your own design - even if you are familiar with previous versions of the program. There are many new features which you need to master.

In this tutorial you will learn the following skills:

using the keyboard;

identifying keys and using key combinations;

starting the *PROTEL-AUTOTRAX* program;

loading the DEMO file;

using the drop-down menus;

quitting from *PROTEL-AUTOTRAX*;

expanding and contracting the display;

using the cursor keys and the mouse;

using the cursor keys with the «SHIFT» key;

getting information about the board;

locating components, tracks, vias and pads;

changing setup options.

Using the keyboard

Throughout this manual a number of conventions have been used to help you decide what to type. One convention is that when a particular key is referred to, it will be enclosed in chevrons. Thus if you are asked to type the key labelled **A** it will be enclosed in chevrons thus, «A». If you are asked to hold down the key marked **SHIFT** it will be referred to as «SHIFT», and so on.

The main section of the keyboard comprises a set of letter keys and number keys as well as some special keys. The one you need to become familiar with immediately is the key labelled «ENTER» or sometimes «RETURN» or sometimes simply an arrow. This key is often used to signal that you have finished entering information.

Some keys are used in conjunction with one other key. These are the «CTRL», «SHIFT» and «ALT» keys. Often you will be asked to type something like **«CTRL» «A»**. This means hold the «CTRL» key down with one finger and press the «A» key with another. Another important key is «BACKSPACE» which deletes the character to the left of the cursor.

Keyboards vary considerably in their layout, but most have a set of **function keys** which are labelled **F1, F2,** etc. Sometimes these are on the left hand side of the keyboard and sometimes they form a line along the top. The use of function keys is optional with *PROTEL-AUTOTRAX* but they can speed up operation considerably.

On the right hand side of the keyboard there is a numeric keypad. Some keyboards also have a set of cursor keys (the keys with arrows on them) and some special keys between the main part of the keyboard and the numeric keypad. The arrow keys will be referred to as «LEFT», «RIGHT», «UP» and «DOWN». Other keys which will be used often are those marked «HOME», «END», «PGUP» and «PGDN». The «DEL» key is used to delete the character which is currently highlighted.

If you have an 'AT'-style keyboard with the extra set of cursor keys between the main keyboard and the numeric keypad, then it is possible that the «SHIFT» key will not work in conjunction with these cursor keys.

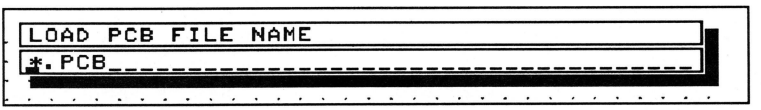
If you are using a mouse then the left button will be referred to as «LEFT MOUSE» and is similar in operation to the «ENTER» key. The right button will be referred to as «RIGHT MOUSE» and is similar in operation to the «ESC» key. This usually takes you back one step or cancels an operation.

Loading the DEMO PCB

If you are using a floppy drive system, insert the *PROTEL-AUTOTRAX* floppy disk marked Program Disk A into drive A: and Program Disk B into drive B:. If you are using a hard drive change to the directory on the hard disk from which you want to run the *PROTEL-AUTOTRAX* programs.

Type: **TRAXEDIT** and then press «ENTER».

The banner screen appears. Press any key to move to the next screen. A window will open in the top left-hand corner of the screen. This is a standard input window which you will see many times while using *PROTEL-AUTOTRAX*.

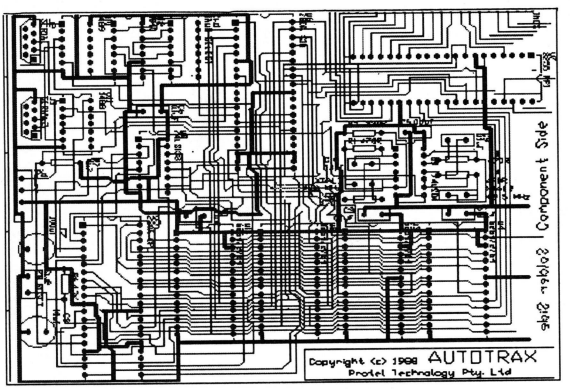


If you are using a floppy drive system, replace Program Disk A with the Library Disk.

Type **DEMO** and then either press the «ENTER» key or click the «LEFT MOUSE» button. The message:

LOADING FILE. PLEASE WAIT.

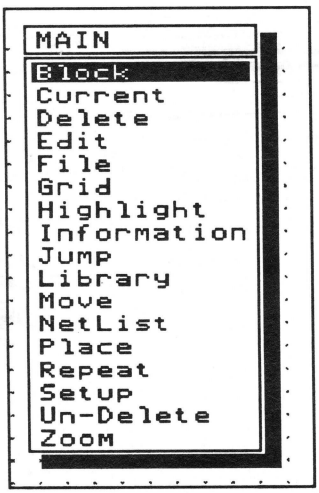
appears, then a diagram showing a simple Z80 microcomputer layout is displayed.



Notice the **status line** at the bottom of the screen which gives important information about the board and the position of the cursor. If you move the mouse or press the cursor keys a few times the numbers which indicate the position of the cursor change.

Using the menus

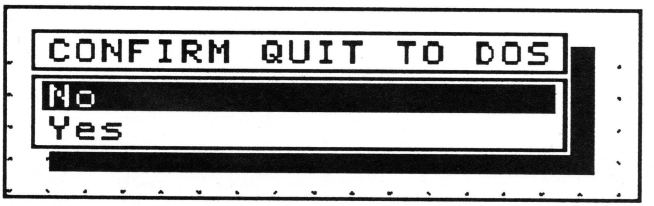
Press the «ENTER» key or the click the «LEFT MOUSE» button and the MAIN MENU window (shown below) appears.



One of the options in the menu is highlighted. This highlight is called the **Selection Bar**. The bar can be moved with the mouse or the «UP» and «DOWN» cursor keys. You can 'activate' or 'select' a highlighted option by clicking the «LEFT MOUSE» button or by pressing the «ENTER» key.

It is important to know how to get out of the program at this stage. Select the ***File*** option and then select ***Quit***. Notice the convention used here. If you are asked to select an option from a menu the menu option will be listed in italics with the first character in bold type. Thus a request to select ***File Quit*** means choose the ***File*** option, then choose the ***Quit*** option.

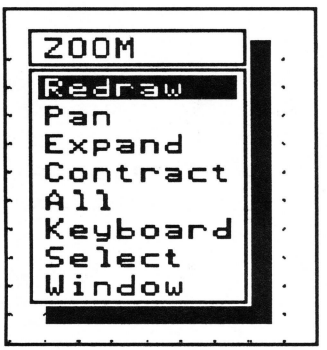
A window opens which asks you to confirm that you want to quit to DOS.



Choose the *Yes* option if you do want to quit. Choose *No* to continue with this tutorial.

The demo circuit is too small to work with easily, so it needs to be expanded. Choose *Zoom* from the MAIN MENU. Remember to make sure that it is highlighted, then click the «LEFT MOUSE» button or press the «ENTER» key to select the option.

The following window appears:

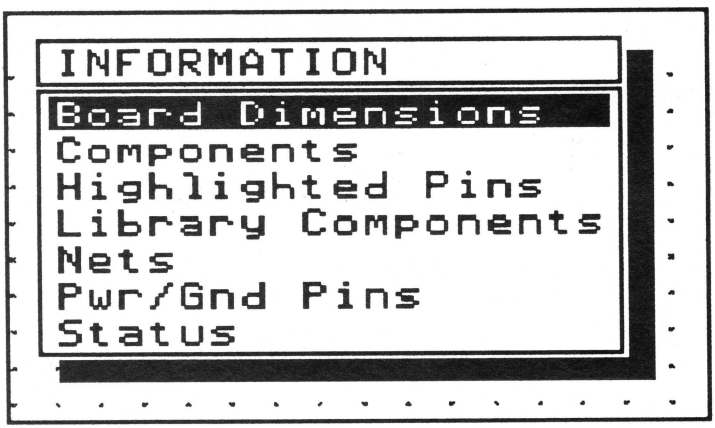


Select *Expand*. The screen is redrawn and the diagram shown in more detail. Select *Zoom Expand* once again.

Now move the mouse until the cursor is in the bottom left hand corner. Alternatively, use the cursor keys. If you hold the «SHIFT» key down while pressing a cursor key on the numeric keypad, the cursor will move faster. Try it.

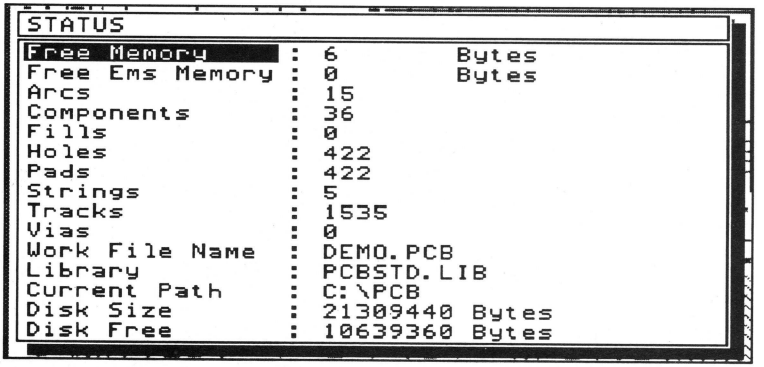
As the cursor moves towards the bottom left hand corner, notice that the screen may be redrawn once or twice. Remember that the cursor position is displayed on the status line.

Select the MAIN MENU, then select *Information*. The following window appears:



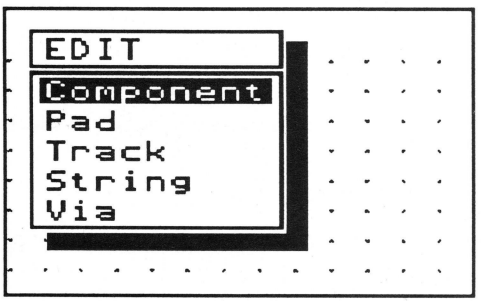
Choose the *Status* option from the *Information* window. Notice that information is provided about the number of components, pads, holes, arcs, fills, strings, tracks and vias. Each of these terms are described in more detail later. You will also notice that information is provided about the name of the current work file, the current path to the directory where the work files are stored and the name of the currently selected library. You can also check the amount of disk space that is available.

Click «RIGHT MOUSE» or press «ESC» to exit from the *Status* window.

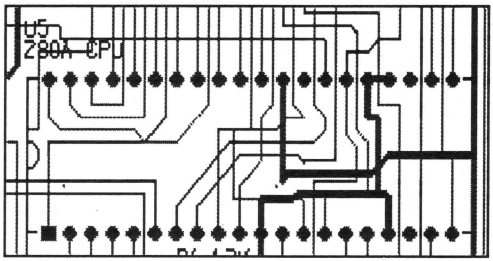


Now try each of the other options in the *Information* window to determine the board's dimensions, the components on the board, the unconnected pins and so on. Press «ESC» or click the «RIGHT MOUSE» button to exit from this menu.

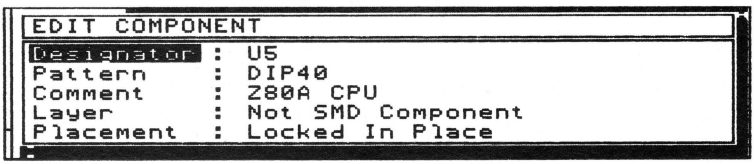
You can find more information about a particular component by using the *Edit* menu. Choose *Edit* and the following menu appears.



Now choose *Component*. The message on the **prompt line** (just below the status line) prompts you to select a component. Move the cursor until it is positioned on the Z80 chip marked U5 (approximately X:1200 Y:1200). The component should look like the one in the following diagram.



Now click «LEFT MOUSE» or press «ENTER» and a window opens showing the component designator, the library name (pattern), a comment and information about the layer and placement type.

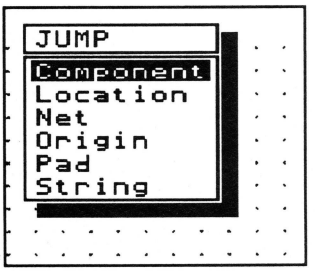


Click «RIGHT MOUSE» or press «ESC» to remove the window. Notice that the:

SELECT COMPONENT

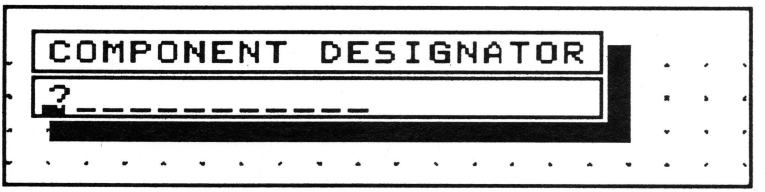
prompt still appears on the prompt line. Click the «RIGHT MOUSE» button or press «ESC» again to exit from the *Edit* option.

Now choose the ***Jump*** option from the MAIN MENU. You can choose to jump to a place on the board by specifying a component, a location, a net, the origin, a pad or a string.

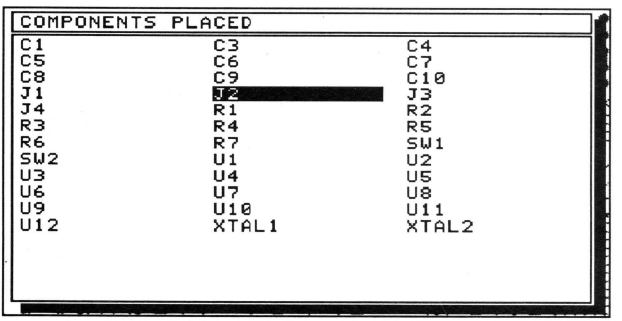


This option will be extremely useful in the future when you are working with a large board.

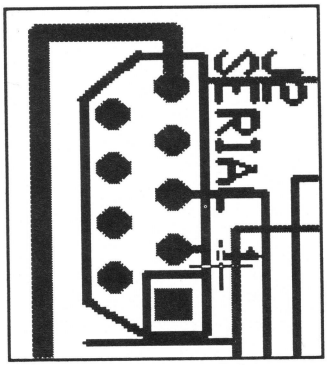
The *Jump* option is also useful if you wish to identify the pin numbers for a component. Choose *Jump Component* and you are asked to select a component. Notice that there is a question mark (?) on the left hand side of the window. This indicates that there is an information window available which will provide a list of components held in the memory of the computer.



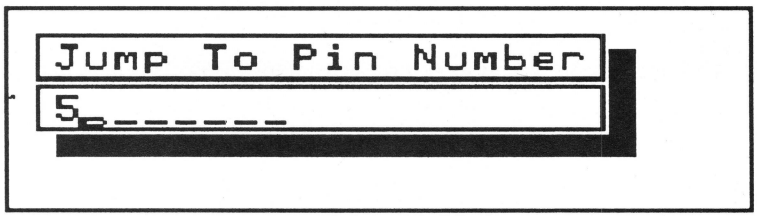
Click «LEFT MOUSE» and the window with a range of choices appears.



Move the selection bar to highlight the DB9 connector labelled J2 then press «ENTER» or click «LEFT MOUSE».



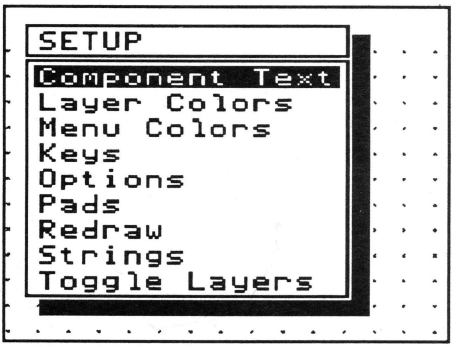
The cursor moves to that component on the board and is situated at pin 1. Now choose *Jump Pad.* You will be asked to select a component. Since the cursor is already over the required component, press «ENTER» or click «LEFT MOUSE». A window opens at the top of the screen prompting you to enter a pin number.



Enter a pin number such as 5 then press «ENTER» or click «LEFT MOUSE». Notice that the cursor moves to that pin. The window stays open allowing you to try other pin numbers. Press «RIGHT MOUSE» or «ESC» to exit from this window.

Now choose the *Setup* option from the MAIN MENU. This menu allows you to change layer colors, menu colors, the default pad type and many other factors which affect the layout. Try changing the menu background

color by choosing *Menu Color* then clicking «LEFT MOUSE» or pressing «ENTER» and observing the changes in color. Press «ESC» or click «RIGHT MOUSE» when you have finished selecting colors.



By now you should be familiar with the menu structure of *PROTEL-AUTOTRAX* and some of the features of board layout. No attempt has been made to place components or tracks or to change layers. This will be done in subsequent chapters.

Some special keys

There are some other keys which have a special function in the *PROTEL-AUTOTRAX* program. These are explained below:

«END»	Redraws or 'refreshes' the screen.
«HOME»	Centers the screen about the current cursor position.
«PGUP»	This key has the same effect as choosing *Zoom Expand.*
«PGDN»	This key has the same effect as choosing *Zoom Contract.*
«*»	This key is on the right of the keypad. Sometimes it also has «PRTSC» on the same key. It is used to swap between the top layer and the bottom layer. The effect can be observed on the status line.

«-»	This key is also on the keypad. It is used to cycle through all the currently selected layers.
«+»	This key is also on the keypad. It is used to cycle through the currently selected layers in the opposite direction to the «-» key.

Speeding up menu selections

Before finishing this tutorial you should be aware of a feature of *PROTEL-AUTOTRAX* which helps speed up menu selections once you become familiar with the menu structure.

Each menu option can be chosen by simply pressing the first key of the option to be selected. Thus to choose ***Jump Pad*** you only need to type **J** then **P**.

You should now quit from the program by choosing ***File*** from the MAIN MENU, then choosing ***Quit*** (or by pressing **F** then **Q**) as described near the beginning of this tutorial.

3 TUTORIAL: LAYING OUT A PCB

In this tutorial the most important aspects of placing components, using the library, laying tracks and using the powerful router will be covered. The tutorial will lead you through the set of steps necessary to begin designing an RS-232 splitter board. As there is insufficient space in the manual to describe laying out the whole board, you will simply place a DB25 connector and the first of the 16 pin chips as well as some by-pass capacitors and a resistor.

In this tutorial you will learn the following skills:

clearing the work space;

choosing a component from the library;

placing a component;

rotating a component;

assigning component designators and comments;

identifying pins on components;

placing tracks manually;

switching layers;

saving a board;

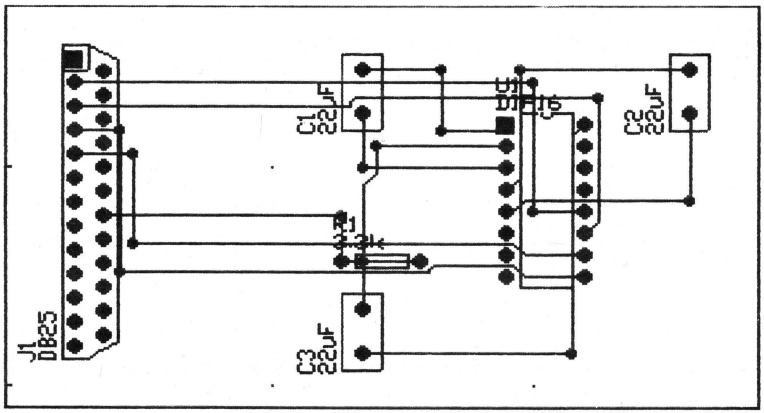
using the pad to pad router;

creating a netlist;

using the automatic router.

Getting started

The aim of this tutorial is to lay out the section of an RS-232 splitter board shown below.



There is one DB25 connector, one DIP16 chip, three capacitors and one resistor on the board. Your first task is to place these.

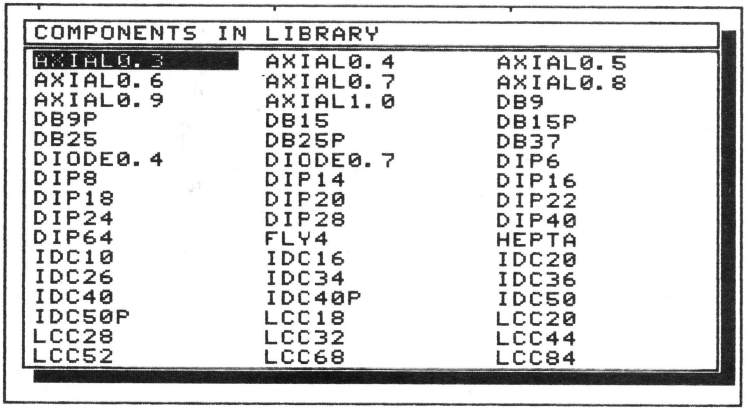
You must first make sure that the board is clear. Either restart the *PROTEL-AUTOTRAX* program or choose ***File Clear Yes***. If you start the program again then press «ESC» or click the «RIGHT MOUSE» button in response to the prompt 'LOAD PCB FILE NAME'.

Placing components

Now press «PGUP» three times and choose ***Place Component***. An input window with the heading 'NAME IN LIBRARY' opens asking for the name of a pre-defined component. Notice that there is a question mark on the input line. Press «ENTER» or click the «LEFT MOUSE» button and a list of the components in the library is displayed.

If you know the name of the component then you could have simply typed it when the input window opened. In this case you would have entered the name **DB25** and pressed «ENTER» or clicked «LEFT MOUSE».

For the purposes of this tutorial, however, select the component from the library list as displayed below.



Move the selection bar to the component named DB25. Press «ENTER» or click the «LEFT MOUSE» button. A window opens and you are prompted to supply a component designator. The question mark in the window means that if you press «ENTER» or click «LEFT MOUSE», the program automatically assign a designator starting with A1, then A2 and so on.

A component designator is usually a letter such as U, J, C, or R followed by a one or two digit number. Connectors are often given designators starting with J, so enter **J1** in this case, then press «ENTER» or click «LEFT MOUSE».

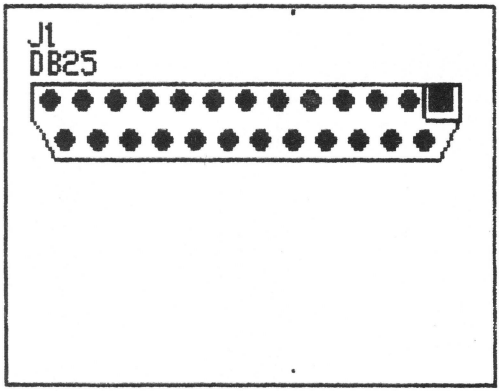
You are now prompted to enter a short comment which appears on the component overlay. Type in **DB25** and an outline of the component appears.

You now see the message:

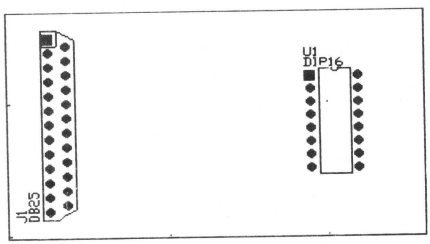
MOVING COMPONENT

on the prompt line at the bottom of the screen.

This means that you can now adjust the position of the component. Press «SPACE» to rotate the component through 90 degrees, then move the white outline (with the cursor keys or the mouse) close to the edge of the board - but not so close that you cannot see the whole component. Try X:250 Y:2500 for example. When the outline is in the correct position, press «ENTER» or click «LEFT MOUSE» and the component is placed.



The next step is to place a 16-pin chip near the J1 connector. At this stage you are still in ***Place Component*** and the input window has the component name DB25 in it, since that was the last component chosen. Enter a question mark and the string DB25 disappears. Press «ENTER» or click the «LEFT MOUSE» button and the information window opens. Choose the component labelled DIP16 then press «ENTER» or click «LEFT MOUSE». Enter the designator **U1** and the comment **DIP16**. Press «ENTER» or click «LEFT MOUSE» after each entry. The outline of the component now appears. Use the cursor keys or the mouse to move it to X:1900 Y:2200. Once again, press «ENTER» or click «LEFT MOUSE» to fix the component in place.



Now you need to place three capacitors. The component you require this time is called RAD0.2. Enter a «?» in the input window. It is not listed in the information window so press the «PGDN» key to see the next screen full of names. Choose RAD0.2 and supply the designator **C1** and the comment **22uF**. Press «ENTER» or click «LEFT MOUSE» to enter each of these items. Press «SPACE» to rotate the outline through 90 degrees,

then move it to X:1350 Y:2250. Press «ENTER» or click «LEFT MOUSE» to place it.

The input window is still showing the name RAD0.2 so simply press «ENTER» or click «LEFT MOUSE» to choose that name for the second capacitor. The designator automatically becomes C2 and the comment is still 22uF. Rotate the second capacitor and place it on the other side of the DIP16 chip at X:2800 Y:2250. Now choose RAD0.2 a third time. Rotate it and move it to X:1350 Y:1150. Click «LEFT MOUSE» or press «ENTER» to fix it in position.

Now move to a point just above C3 (say, X:1250 Y:1550) and choose the component labelled AXIAL0.3. Assign the designator **R1** and the comment **3.3K.** Press «ESC» or click «RIGHT MOUSE» to exit from the *Place Component* option.

The display might be a bit messy by now so press the «END» key to redraw the screen.

Laying tracks

Now you are almost ready to start laying tracks. Before you can, you need to know the pin numbers of the components. In the previous tutorial you learned how to identify pin numbers using *Jump Pad.* Use this option to find pin 1 on J1, U1, R1 and C1. From this information you can work out most of the others. It might also be necessary to locate pin 25 on J1 and pin 16 on U1.

The following table provides a list of the pins which must be connected.

U1-1 to C1-2

U1-16 to C3-1

U1-11 to J1-3

U1-9 to J1-4

U1-4 to C2-2

J1-20 to R1-1

U1-3 to C1-1

U1-12 to J1-2

U1-10 to J1-5

U1-5 to C2-1

U1-2 to C3-2 to R1-2

The first entry in the table indicates that you must connect pin 1 on the component U1 to pin 2 on the component C1. The entry under it shows that you must connect pin 16 on the component U1 to pin 1 on the component C3.

This information becomes important later as it is the basis of the concept of a **Netlist**. A netlist is a list of all connections on the board and is used to make connections automatically with the *Netlist Route* option. Each group of connections in a netlist (e.g. U1-1 to C1-2) is called a **Net**.

Just in case anything goes wrong, save the board. To do this choose ***File Save***, supply a filename, then press «ENTER». Choose ***Yes*** to confirm the fact that you want to save the PCB file.

Try connecting the first couple of pads manually. Press the «*» key on the keypad to change from the component side (top layer) to the solder side (bottom layer). On the status line the layer indicator message changes from 'L: Top Layer' to 'L: Bottom Layer'. Choose ***Place Track*** and the message:

SELECT TRACK START POINT

appears on the prompt line at the bottom of the screen. Move the cursor to pin 1 on U1 (it is a square pad) - you are going to connect it to pin 2 on C1. Click «LEFT MOUSE» or press «ENTER». The message:

PLACE TRACK

appears on the prompt line.

Now move the cursor towards C1 keeping the line horizontal. When the line is half way towards C1 (about X:1650 Y:2200) click «LEFT MOUSE» or press «ENTER». Press «*» to change layers. Now move the line vertically until it is in line with pin 2 on C1 (about X:1650 Y:2450). Click again or press «ENTER» then press «*» to change to the bottom layer. Move the cursor to pin 2 and click again or press «ENTER».

A connection now extends from U1-1 to C1-2. Notice that this last section has been placed on the bottom layer and that a via has been used to change layers. Press «ESC» or click «RIGHT MOUSE» twice to exit from the track laying routine.

Using the router

Manually laying tracks is a very time consuming process - especially when you have to change layers all the time. This is obviously very cumbersome. The main advantage is that you have total control over the placement of tracks.

The *PROTEL-AUTOTRAX* program has an automatic track laying option built-in. This process is known in the industry as **routing**. The router will connect two pads which you point to, or it will use a netlist to make some or all of the connections on the board. This is a very powerful feature of the *PROTEL-AUTOTRAX* program and is described in detail in chapter 20.

Now let the *PROTEL-AUTOTRAX* router lay a track for you. Choose *Netlist Route Pad to Pad.* The message:

ROUTE: SELECT FIRST PAD

appears. You need to connect U1-3 to C1-1, so move to pin 1 on C1 and press «ENTER» or click «LEFT MOUSE».

The message now changes to:

ROUTE: SELECT SECOND PAD

Move to pin 3 on U1 and press «ENTER» or click «LEFT MOUSE».

The router now takes over and determines a path for the tracks. It is quite likely that it will also place a via to route that track to the other side of the board.

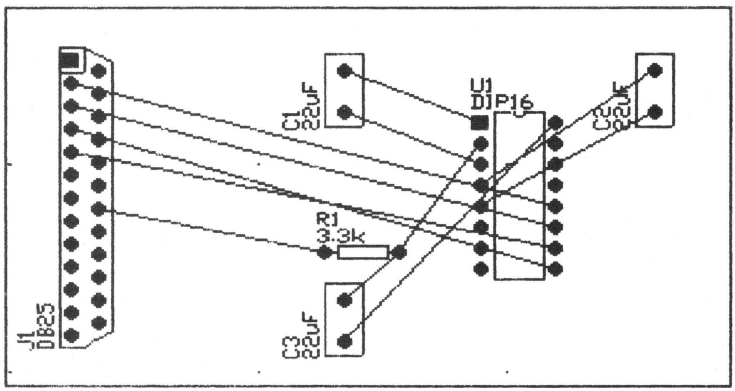
Now route all the other connections in the list above. When you have finished, press «ESC» or click «RIGHT MOUSE» to escape from the router and netlist menus.

To do an automatic route you need a netlist which contains information on all connections to be made. This is normally done with the *PROTEL-SCHEMATIC* program, but there is an option which allows you to produce a netlist of a finished board within the *PROTEL-AUTOTRAX* program.

Choose *Highlight Make Netlist.* You are asked to supply a name for the netlist. Type in the name **NET232** and press «ENTER». The board is then scanned and all connections are stored in the netlist file. This file is used for automatic routing.

Now save the board giving it a different name and recall the original unrouted board. Choose *Netlist Get Nets*. You are asked to supply the name of a netlist file. Enter **NET232**, then press «ENTER» or click «LEFT MOUSE». The netlist is loaded and optimised and then the *Netlist* menu is displayed. At this stage the netlist has been loaded into the memory and is available for routing.

You can get visual feedback about the netlist by choosing the *Show All Nets* option. A set of thin white lines is displayed showing the connections to be made. This set of connections is called a 'ratsnest' and is, essentially, the information which is used to automatically place the tracks.



You can now examine one of the features of *PROTEL-AUTOTRAX* which can be very useful when laying out large boards, or boards which are difficult to route. It is possible to move a component and to see the effect that the move has on the ratsnest. In this way you can move a component which is in a very dense area to a place where routing will be easier.

Choose *Move Component*. You are prompted to select a component. Select the DIP16 chip and drag it. As it moves you can see all the connections in the ratsnest moving as well. Now put the component back in its original spot and press «ENTER» or click «LEFT MOUSE» to exit from the *Move* option.

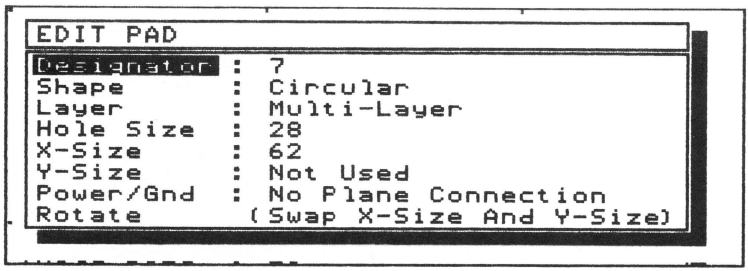
Choose *Route Board*. The routing now takes place automatically using the information from the netlist. Notice that on the status line

information is provided about the type of connection being made. This is described in more detail in the routing chapter.

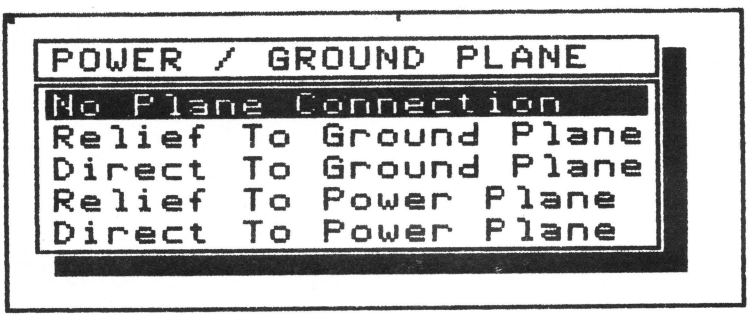
Connecting to power and ground planes

The final connections you have to make are to the power plane and ground plane. Pin 7 on the DB25 connector and pin 15 on the DIP16 chip have to be connected to the ground plane. Pin 16 on the DIP16 has to be connected to the power plane. (This is assuming, of course, that you are going to construct a board with power and ground planes.)

Choose *Edit Pad.* You are prompted to select a pad. Select pin 7 on the DB25 and a window opens displaying information about the pad.



Notice that there is an option which shows the power and ground status. Choose this option and another window opens listing the possible connections. Choose *Relief to Ground Plane.*



Press «ESC» or click «RIGHT MOUSE» and you are prompted to select another pad. Now choose pin 15 on the DIP16 and make the same connection. Finally, choose pin 16 on the DIP16 chip and connect it to the power plane by choosing the ***Relief to Power Plane*** option.

You have now completed the section of the board shown in the first diagram in this chapter. If you want to save your work, choose ***File Save*** and enter a filename. Press «ENTER» or click «LEFT MOUSE» and the file is saved to the disk.

4 TUTORIAL: CREATING NEW COMPONENTS

In this tutorial you will learn how to create you own component pattern and how to store the pattern in the library for later use. You may not need to do this for a while because a comprehensive library of component patterns is supplied with the *PROTEL-AUTOTRAX* package. In addition, many components can be created by simply altering existing library patterns using the ***Library Explode*** option.

Nevertheless, you will also learn other skills which will help you become more familiar with the procedure for laying out a board.

The skills you will learn will be:

how to use the snap grid;

relating cursor movement to the snap grid;

laying tracks of a given length;

creating new pad types;

changing the currently selected pad type;

placing pads;

using the ***Repeat*** options;

placing arcs;

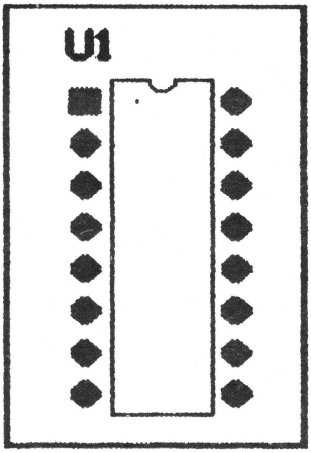
defining a block;

creating a new library;

adding a component to the library.

Getting started

The component that you are going to create in this tutorial is a simple DIP16 pattern (which is already in the standard library - but the exercise is useful).



Before you make any changes, either restart the *PROTEL-AUTOTRAX* program or choose ***File Clear Yes***.

The pattern you are going to create is laid out on a 25 mil grid. The track segments are 800 mils, 75 mils or 50 mils. The radius of the arc which makes up the 'notch' in the chip is 25 mils, and the pads are placed 100 mils apart (measured from center to center).

Choose ***Grid Snap Grid*** and make sure the size of the grid is 25 mils. This means that the cursor will be constrained to move in increments of 25 mils.

You will define two types of pad for this pattern. Pin 1 is a square pad of side 56 mils. The rest are circular with a diameter of 56 mils. Both of these pad types need to be defined because they are not standard - they have just been invented for the purpose of this exercise.

Choose ***Setup Pads New***. A window opens with the heading 'PAD TYPE NAME'. Enter **SQUARE56**. The 'EDIT PAD TYPE' menu appears. Choose ***Shape***, then choose ***Rectangular***.

Now choose **X**-***Size*** and change the size to 56 mils. Choose **Y**-***Size*** and change the size to 56 mils. The default hole size of 30 mils is

satisfactory. Press «ESC» or click «RIGHT MOUSE» to exit this option. The square pad is now defined.

Repeat the procedure and define a pad called **ROUND56** which is circular with a diameter of 56 mils and also a hole size of 30 mils. Remember to press «ESC» or click «RIGHT MOUSE» twice to exit this option.

The next step is to make sure that the circular pad called ROUND56 is the currently selected pad and that the track width is set to 12 mils.

Choose *Current Pad Type* and select ROUND56. Also check that the track width is 12 mils.

You are now ready to start creating the component.

Placing some tracks

The tracks for this component will be placed on the top (component) overlay, so either choose *Current Layer Top Overlay* to change to that layer, or press either «+» or «-» on the keypad until the overlay is the current layer.

You are now ready to lay the tracks which will make up the inner part of the component.

Move to X:1000 Y:1000 and press «PGUP» four times. Choose *Place Track*. Make sure the cursor is on X:1000 Y:1000 then press «ENTER» or click «LEFT MOUSE» to 'SELECT TRACK START POINT'.

Move the cursor across to X:1200 Y:1000 then press «ENTER» or click «LEFT MOUSE». A short track is drawn. Now move the cursor up to X:1200 Y:1800 and press «ENTER» or click «LEFT MOUSE». A track which forms the right side of the outline appears. Now move across to X:1125 Y:1800 and repeat the process.

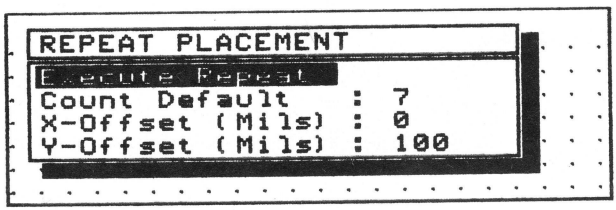
Now you have to leave a gap for the notch, so press «ESC» or click «RIGHT MOUSE» once and then move to X:1075 Y:1800. Press «ENTER» or click «LEFT MOUSE» then lay a track to X:1000 Y:1800. Now complete the outline by laying a track down to X:1000 Y:1000.

Press «ESC» or click «RIGHT MOUSE» twice to finish laying tracks.

Placing pads and using *Repeat*

You are now ready to place the pads so choose *Place Pad.* The message 'PLACE PAD' appears on the prompt line at the bottom of the screen. Move the cursor to X:1250 Y:1050 and press «ENTER» or click «LEFT MOUSE». A pad appears on the board. The shape of the pad is determined by the shape chosen for the currently selected pad in *Current Pad.* You are then prompted to place another pad. Even though there are more to place, press «ESC» or click «RIGHT MOUSE» to exit the *Place Pad* routine. Because you have several pads to place, you can save time by using the *Repeat* command.

Choose *Repeat* from the MAIN MENU and the following window opens.

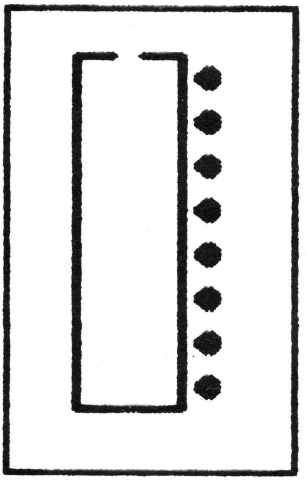


Each time an item is placed on the board, the information about the latest placement is stored so that the action can be repeated. The window contains the information which the program needs to repeat the action.

You need to place seven more circular pads. Choose *Count Default* and change it to 7. Since the pads are to be placed in a vertical line 100 mils apart, change the X offset to 0 and the Y offset to 100 mils.

Now choose *Execute Repeat Yes.* Notice that seven pads have been laid. Your previous *Place Pad* instruction has been repeated seven times and each one is 100 mils apart.

Your component should now look like the one in the diagram below.



Now choose ***Place Pad.*** Move to X:950 Y:1050 and press «ENTER» or click «LEFT MOUSE». Another pad appears. Choose ***Repeat Execute Repeat*** again. Change the count to 6 and another six pads will be placed.

Now you are ready to place pin 1 which is a rectangular type, so choose ***Current Pad*** and change the type to SQUARE56. Move to X:950 Y:1750 and choose ***Place Pad*** to place the rectangular pad. Now you can see the bottom part of the pattern and the two rows of pads.

Placing an arc

Now you need to create the notch at the top of the chip. Move to X:1100 Y:1800 and choose ***Place Arc***. Change the radius to 25 mils by using the «RIGHT» arrow key to increase the radius and the «LEFT» arrow key to decrease the radius. Press «ENTER» or click «LEFT MOUSE» to confirm the size. The message 'SELECT ARC SEGMENTS' appears on the prompt line.

You only want the bottom half of the circle to appear. To do this press the «UP» arrow key and then the «LEFT» arrow key and the top part disappears. Press «ENTER» or click «LEFT MOUSE». The notch will now appear.

Press «ESC» or click «RIGHT MOUSE» to exit from this option.

You are almost ready to add the component to the library. The only remaining task is to assign the pin numbers or designators. The rectangular pin will be pin 1, the one under that will be pin 2, and so on. The pin in the top right hand corner will become pin 16.

Choose ***Edit Pad*** and then select the rectangular pad. Now choose ***Designator*** and assign the value 1. Press «ESC» or click «RIGHT MOUSE». Now select each of the remaining pads and assign their designators.

Adding the component to the library

The next step is to define the new component as a block by choosing the ***Block Define*** option. When you choose the option the message 'SELECT FIRST CORNER' appears on the prompt line at the bottom of the screen. Move the cursor to X:850 Y:1900 and press «ENTER» or click «LEFT MOUSE».

Now the prompt changes to 'SELECT SECOND CORNER'. Move the cursor to X:1350 Y:950 and press «ENTER» or click «LEFT MOUSE».

The component should now be highlighted indicating that it is a defined block. The prompt 'SELECT REFERENCE POINT' is now displayed on the prompt line. The reference point is that part of the component that will be placed at the cursor position during a later ***Place Component*** operation. Choose the center of the rectangular pad (i.e. pin 1) as the reference point.

Since you probably do not want to add this component to your standard library, create a new library to store the component in. Choose ***Library New Library*** and enter the name **NEW** for the library.

Now choose ***Library File*** and nominate NEW as the currently selected library. You can check that it is the currently selected library by choosing ***Information Status.***

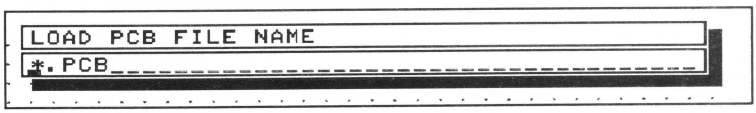
Choose ***Library Add.*** You are asked to enter a name for the new component. Enter a name and then press «ENTER» or click «LEFT MOUSE». Choose ***Yes.*** The component is added to the library.

If you now choose ***Library List*** you will see the component listed. It can now be placed on the board just like any other library component.

5 TUTORIAL: USING THE LINE EDITOR

The *PROTEL-AUTOTRAX* package contains a simple, yet effective, line editor which is available in all input windows.

You will have already used an input window when you loaded the DEMO file in a previous tutorial.



You can use the line editor to change text and to correct errors which may arise when typing information into an input window.

This section contains a short tutorial to help you become familiar with the line editor and a summary of the functions available.

The skills you will learn will be:

using an input window;

deleting text at the cursor;

inserting text at the cursor;

moving around the window;

deleting an entire word or line;

exiting from an input window.

The input window explained

The first point to note is that when an input window opens it may already have some information on the edit line. This may either be a 'default' specification such as *.PCB or DIP40 or a question mark.

If a default specification or a question mark appears then clicking «LEFT MOUSE» or pressing «ENTER» will cause a selection window to open. The window will contain some names. One of these can be selected by moving the selection bar to that name and then clicking «LEFT MOUSE» or pressing «ENTER». This procedure overcome the need for you to type information.

Whenever the window first opens, the cursor appears on the left hand side of the edit line. If you want to type a filename or the name of a component rather than selecting one from a list then simply type the name and any text that was already in the window will disappear.

You may sometimes make a typing error when entering a filename or a component designator. Alternatively, you may simply want to make a minor change to the default value already in the window. This is where the line editor can help.

Now choose *File* from the MAIN MENU and then choose *Load*. The input window opens.

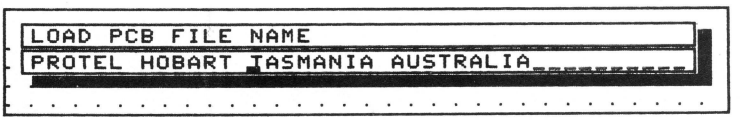
Notice that there is a default specification of *.PCB on the left hand side and that the flashing cursor appears under the asterisk.

Editing a line

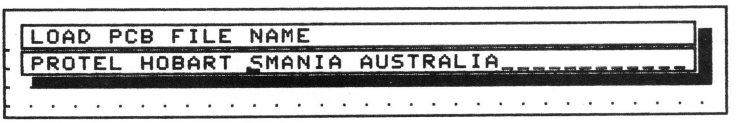
Type the words **PROTEL TASMANIA AUSTRALIA.**

Notice that the cursor is positioned after the **A** at the end of the line. Now type **«CTRL» «A»** twice and note that the cursor has moved back

to the start of the word **TASMANIA**. Now type the word **HOBART** followed by a space. The new text is inserted to the left of the cursor position.



Press the **«DEL»** key twice. The characters under the cursor are deleted.



Now type **«CTRL» «T»** and the rest of the word **TASMANIA** is deleted.

Press the **«HOME»** key and then the **«END»** key and the cursor moves to the beginning of the line then to the end of the line. Press **«HOME»** again, then use **«CTRL» «F»** to move the cursor right one word. Notice that the cursor is positioned under the **H** in the word **HOBART**.

Since you do not want to use the text you have just typed in, press **«CTRL» «Y»** to delete the entire line.

By now you should be ready to alter text in any input window. To help you with the line editor, a full summary of the features is given in the next section.

Line editor functions summary

«HOME»	Move to the beginning of the line.
«END»	Move to the end of the line.
«INS»	Toggle insert mode. Pressing «INS» once switches to overstrike mode. Pressing it again switches back to insert mode.
«ESC»	Quit without changing the line.
«ENTER»	Accept the line.
«DEL»	Delete the character at the cursor.
«BACKSPACE»	Delete the character to the left of the cursor.
«LEFT»	Move the cursor left one character. An alternative is to use «CTRL» S.
«RIGHT»	Move the cursor right one character. An alternative is to use «CTRL» D.
«CTRL» «A»	Move the cursor left one word.
«CTRL» «F»	Move the cursor right one word.
«CTRL» «S»	Move the cursor left one character.
«CTRL» «D»	Move the cursor right one character.
«CTRL» «G»	Delete the character at the cursor position.
«CTRL» «H»	Delete the character to the left of the cursor.
«CTRL» «R»	Restore the line to its state on entry.
«CTRL» «T»	Delete the word to the right of the cursor.
«CTRL» «Y»	Delete the entire line.
«CTRL» «X»	Delete all characters to the end of the line.

6 TUTORIAL: ASSIGNING KEY MACROS

A powerful, and very useful, feature of *PROTEL-AUTOTRAX* is the ability to define your own macros. If you have not heard the term macro before then don't worry. It is simply a term that is used to describe a method of assigning commonly used keystrokes to a single name or key.

The use of macros helps to speed up operations that you need to carry out often.

In this tutorial you will learn the following skills:

clearing previously assigned macros;

defining a new macro;

assigning a macro name;

assigning a macro to a function key;

implementing a macro;

using the record facility to define a macro;

terminating the record facility;

saving a macro;

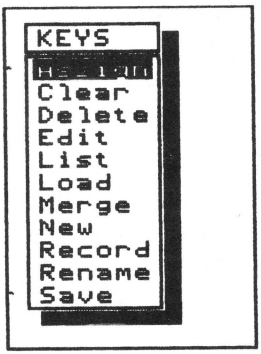
creating nested macros.

Defining a macro

An example will help. When laying out a board you will need to swap from the top layer to the bottom layer from time to time. One way of doing this is by choosing *Current* from the MAIN MENU, then choosing *Layer Bottom Layer*. If you use single keystrokes you would type **C L B**.

Before starting this tutorial you should make sure that there are no macros already defined. Choose *Setup Keys Clear Yes* to clear any macros from the memory.

To assign this sequence of keystrokes to a macro, choose *Setup Keys*. The *Keys* menu is displayed.



Choose *New* to define a new macro. A window opens prompting you to input a name for the macro. The name can be up to 10 characters long. Choose a name that reflects the function to be carried out by the macro. At this stage type in:

BOTTOM

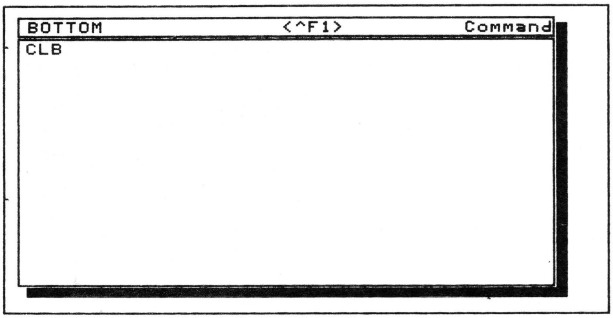
Now you are asked which key to assign the macro to. The program supplies the name of the next available macro key such as «^F1». This stands for pressing the «CTRL» key at the same time as holding down the function key labelled F1.

If you do not want to assign the macro to «^F1» press one of the other function keys. You can also use the function keys in combination with the «SHIFT» and «ALT» keys or by themselves.

Provided the key is available a window opens to allow you to type in the required keystrokes. Type:

C L B

then press the «ENTER» key or click the «LEFT MOUSE» button. Remember that the letters **C L B** stand for the menu selections ***Current Layer Bottom.***



Now press «ESC» twice or click the «RIGHT MOUSE» button twice to finish defining macros. Make sure that you are on the top layer by choosing *Current Top Layer*. To execute the macro, hold down the «CTRL» key and press the function key labelled «F1». Notice that the bottom layer is now selected.

Using the record facility

Another macro which can be quite useful is one which places a pad and then moves to the next pad position. To make sure that you are at the correct *Zoom* level for this exercise, choose *Zoom Keyboard* and then input **5**.

To define the new macro choose *Setup Keys Record*. This option lets you carry out a set of actions and automatically stores your keystrokes in the macro.

A window opens prompting you to enter the 'NEW KEY MACRO NAME'. Enter the name of the macro as **PADS**. You are then asked to specify the key to which the macro will be assigned. Hold down the «CTRL» key and press the function key marked «F2».

Now you will hear a beep and the program is ready to 'record' all your keystrokes.

Choose *Place Pad*. You are prompted to place a pad. Press «ENTER» or click the «LEFT MOUSE» key to place the pad and now press the «RIGHT» cursor key four times. The cursor moves to the right by 100

mils. Press «ESC» or click the «RIGHT MOUSE» button to exit from the *Place Pad* option.

Now, hold down the «CTRL» key and press the «END» key. This turns off the 'recorder' and signifies that you have finished the macro.

Now press «CTRL» «F2». Notice that a pad has been placed and the cursor has moved to a new position ready to place another pad. Press «CTRL» «F2» again and notice that another pad has been placed.

Using nested macros

A powerful feature of macros is that one can call another. This is sometimes referred to as 'nesting' macros.

An example for you to try is creating a pin array consisting of four rows of pads each of which has five pads. You should make sure that the pad laying macro which you assigned to «CTRL» «F2» is still there.

Choose *Setup Keys New* and input the name **Line**. You are prompted to supply the name of the function key. Overwrite the default value by typing «CTRL» «F3» then pressing «ENTER» or clicking «LEFT MOUSE».

In the macro window type «CTRL» «F2» five times. This produces a line of five pads. Then press «DOWN» «DOWN» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» «LEFT» to return the cursor to a point below the first pad. Terminate this macro definition by pressing «ENTER» or clicking «LEFT MOUSE».

Now define another macro and call it **Array**. Assign this definition to «CTRL» «F4». Now press «CTRL» «F3» four times. Terminate the definition by pressing «ENTER» or by clicking «LEFT MOUSE».

Now press «ESC» twice to return to the PCB. Pick a blank area on the board and press «CTRL» «F4». The macro executes automatically and lays down the array of pads.

7 TUTORIAL: USING THE *TRAXPLOT* PROGRAM

This tutorial will show you how to do a simple draft print and a plot of the sample PCB file. *PROTEL-AUTOTRAX* has a flexible set of output routines which allow output to be sent to draft printers, plotters, photoplotters and disk files.

In this tutorial you will learn the following skills:

loading the *TRAXPLOT* program;

interpreting the menu levels;

interpreting the status screen;

loading a .PCB file;

interpreting the *Information* window;

setting up the parameters for a print;

setting up the parameters for a plot;

producing a draft print;

producing a draft plot.

Starting the program

If you are using a dual floppy disk system, put the *Plot* disk into drive A: and the disk with the .PCB file that you want to print into drive B:.

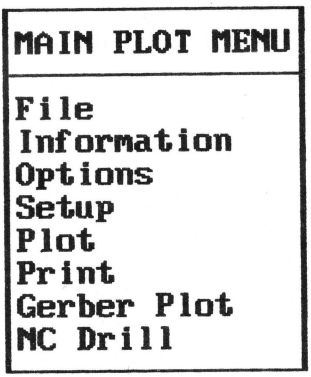
If you are using a hard disk drive you should make sure that you are in a directory from which the *TRAXPLOT* program is accessible.

To start the *TRAXPLOT* program simply type:

TRAXPLOT then press «ENTER»

at the DOS prompt.

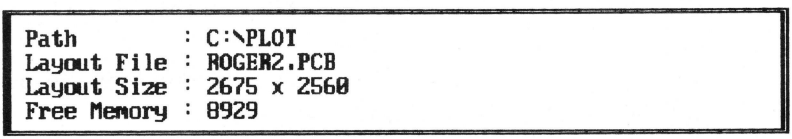
You will see the following menu.



The MAIN PLOT MENU is displayed on the left side of the screen and a status window is displayed at the bottom of the screen.

Choose *File Load* and enter the file name DEMO in the input window. An information window opens showing the number of pads, tracks, components, etc. that are being loaded. A second pass shows the percentage of the board which has been analysed.

When the file has been loaded, the status screen is updated to show the name of the .PCB file, the dimensions of the board and the amount of free memory remaining. The path name of the directory is also displayed.



The *File* menu is still displayed. Press «ESC» or click «RIGHT MOUSE» to return to the MAIN PLOT MENU.

A draft print

Now choose *Options Type of Plot* and choose *Check Plot* from the 'Select Plot Type' window. Also check that the options selected for border, pads and strings are 'on'. Press «ESC» or click «RIGHT MOUSE» to exit from this menu.

Lets assume that you have an HP Laser Jet II printer connected to a serial port of your computer.

Choose *Setup Printer*. Now check that *Type* has been set to 'HP Laser Jet II'. Choose *Device* and another window opens listing the options you can choose for the port to connect the printer to. Choose 'Serial Port 1'. Check the other options to make sure they are sensible. Press «ESC» or click «RIGHT MOUSE» twice to return to the MAIN PLOT MENU.

PRINTER SETUP		
Type	:	HP LASER JET II
Device	:	Serial Port 1
Scale	:	1.00
X Correction	:	1.00
Y Correction	:	1.00
X Offset	:	0
Y Offset	:	0
Orientation	:	Auto Centre
Quality	:	Draft
Paper	:	Continuous

Check that your printer is connected to the port you have selected and make sure that it is online. Now choose *Print* and the draft print of your board will begin to appear.

Plotting a board

Now you can use a plotter to produce a check plot. Choose *Options Type of Plot* and choose *Check Plot* from the 'Select Plot Type' window. Also check that the options selected for border, pads and strings are 'on'.

Press «ESC» or click «RIGHT MOUSE» to exit from this menu.

Let's assume that you have a DMPL plotter connected to a serial port of your computer. (If you haven't, then choose from the list provided in *Setup Plotter Type*).

Choose *Setup Plotter*. Now check that *Type* has been set to 'DMPL'. Choose *Device* and another window opens listing the options you can choose for the port to connect the printer to. Choose 'Serial Port 1' by moving the selection bar to that option then pressing «ENTER» or clicking «LEFT MOUSE». Check the other options to make sure they are sensible. Press «ESC» or click «RIGHT MOUSE» twice to return to the MAIN PLOT MENU.

```
PLOTTER SETUP

Type          :              DMPL
Device        :     Serial Port 1
Scale         :              1.00
X Correction  :              1.00
Y Correction  :              1.00
X Offset      :                 0
Y Offset      :                 0
Orientation   :            Normal
Quality       :             Draft
Pens
```

If you now choose *Information*, a window opens which provides you with information about the plot type, the scale and the size of the board. You cannot change anything in this window - it is for information only. Press «ESC» or click «RIGHT MOUSE» to exit from this window.

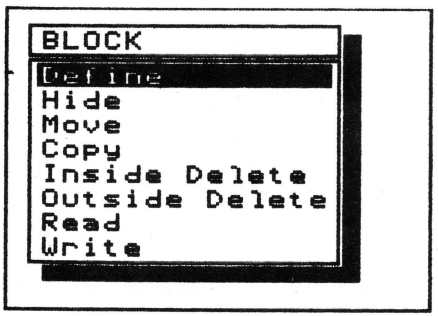
If you are happy with the setup options, choose *Plot* from the MAIN PLOT MENU and your check plot will appear on the plotter.

REFERENCE SECTION

8 MANIPULATING BLOCKS

When you are laying out a printed circuit board, it quite often happens that you need to manipulate a block of components. For example, you may want to copy a block of components, move some components or transfer a block from one PCB to another.

The options in the ***Block*** menu allow you to manipulate blocks quickly and efficiently. They are a powerful feature of the *PROTEL-AUTOTRAX* package and can greatly speed up board layout. You can define a block, copy a block, move a block, hide a block, load or save a block and delete a block.



Defining a block

Before you can carry out block operations such as moving, copying and deleting, you must first define the block.

Choose the ***Define*** option from the ***Block*** menu. The following prompt appears on the status line:

SELECT FIRST CORNER

Move the cursor to one corner of the block you want to highlight and click the «LEFT MOUSE» button or press «ENTER». The prompt changes to:

SELECT SECOND CORNER

Move to the diagonally opposite corner and click the «LEFT MOUSE» button or press «ENTER».

The block is highlighted and the prompt:

SELECT REFERENCE POINT

appears.

The reference point is any point you decide to choose to assist you in moving and placing the block. When you move or copy a block, it is the reference point which moves with the cursor. Some people make one of the corners the reference point, but it is also possible to make the corner of a particular component in the block a reference point. If you are marking a block in order to create a library component, it is a good idea to select the pad which will become pin 1 as the reference point. The reference point can also lie outside the block.

If you want to define a new block and there is one already defined, don't worry! Simply go ahead and define the new block and the previous block is released.

'Hiding' a block

Use ***Block Hide*** when you want to take the highlight off a particular block but do not want to define a new block.

The ***Hide*** option simply removes the highlight from a block. You will notice that the highlight disappears from the selected block once you choose ***Hide*** indicating that the block is no longer defined..

Moving a block

Once you have defined a block with ***Block Define***, you can move it to another place on the board.

Choose ***Block Move*** and the prompt

PLACE BLOCK

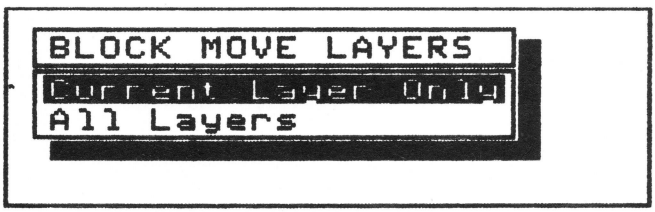
appears. You can now specify the new position for the block. Use the mouse or the cursor keys to position the outline of the block in the

required place. Remember that the cursor position dictates where the reference point will be placed.

Sometimes you may want to place the block at a different orientation to the original. There is a summary of the keys which can be used to rotate, flip and move the block during a *Move* operation on page 66.

When you place the block, a menu appears asking if you want the current layer only or all layers.

If you simply want to drag the tracks on the current layer then choose *Current Layer Only*. With this option no pads or vias are dragged since they belong to every layer. If you wish to affect all tracks, then choose *All Layers*.



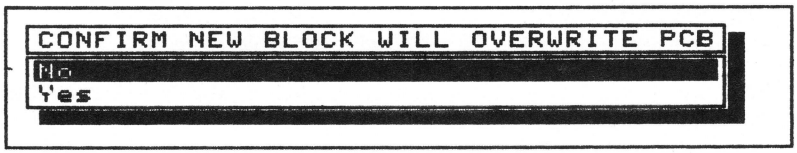
Click the «LEFT MOUSE» button or press «ENTER» to position the block.

The block is then moved. A delay may occur while the tracks are sorted and dragged to the new position. Don't worry if they appear to be messy - they can be sorted out later.

An important point to note is that if a component only lies partly within the defined block it will not be moved. A component must be completely enclosed by the highlighting. If it is not, you will need to redefine the block. However, if the associated text such as the component designator and comment do not fall within the block outline, you do not need to redefine the block. The text is moved with the component regardless of whether it falls within the block or not.

If a pad centre or arc centre falls within the block it is moved. A filled area must be completely enclosed. A track which is completely within the block is moved with the block. If only one end of the track lies within that block then the track is stretched to maintain the connection.

Before a *Move* operation is performed the program checks the limits of the existing board. If the block is to be moved to a place within that area, the following window appears:



Choose *Yes* to overwrite the board or else choose *No* and move the block outline to a different place on the board.

The following keys and buttons help you place the block in any orientation.

«SPACE»	Rotates the block 90 degrees. Pressing it twice rotates the block 180 degrees and so on.
«X»	Pressing «X» flips the block in the X direction.
«Y»	Pressing «Y» flips the block in the Y direction.
«LEFT»	Pressing the left cursor key moves the block left.
«RIGHT»	Pressing the right cursor key moves the block right.
«UP»	Pressing the up cursor key moves the block upwards.
«DOWN»	Pressing the down cursor key moves the block down.
Mouse	Moving the mouse in any direction also moves the block in that direction.
«ENTER»	Pressing the «ENTER» key fixes the block in its current position. Pressing «LEFT MOUSE» has the same effect.

Note that these commands also work when you are placing or moving components and strings.

Copying a block

The *Copy* option is similar to the *Move* option (see pages 64-6) with one important difference. When a block is copied a new copy of each of the components in the block is made and the original components remain. The component designators are incremented to the next free value to ensure that the new designator is unique.

Thus if the component with the designator U2 is copied, the new component is given a new designator depending on the number of 'U' designators already on the board. If there were already the designators U1 to U6, the copied component would be labelled U7.

Remember that when copying a block it is possible that you will overwrite an existing part of the PCB. When you *Copy* a block, the new block is highlighted and becomes the defined block.

Deleting a block

Before you can delete a block you must first define it. See pages 63-4 for a description on how to define a block. Once it is defined, you can delete everything in the defined area or everything outside the defined area.

Choose ***Block Inside Delete*** to delete everything inside the defined block. Choose ***Block Outside Delete*** to delete everything outside the defined block. In each case you are asked to confirm that you want to delete the area. Choose *Yes* to confirm, or *No* if you want to change your mind.

Care must be taken with block deletion commands. It is strongly recommended that you save a copy of the file before attempting a deletion.

One point to bear in mind when deleting outside a block is that a track which passes completely through the defined block is also deleted. This happens because the part of the track inside the block is simply a route between connections which lie outside the block.

If you try to delete without having defined a block, the message:

BLOCK NOT MARKED

is displayed. You should then choose ***Block Define*** to define the block you want to delete.

Reading a block

Since there are common blocks of components that you may want to use on many boards, it is possible to store a number of component configurations in separate .PCB files and then read them into the current board when you require them. You can identify a file stored on the disk, then merge it into the board you are working on in the computer's memory.

To read a block, choose ***Block Read*** and specify the name of the .PCB file. The prompt 'PLACE BLOCK' appears. You are asked to confirm the fact that you want to load a .PCB file. Choose *Yes* to load the file.

Move the block outline until it is in the position you require. Remember to use «SPACE», «X» and «Y» to rotate or flip the block, then click «LEFT MOUSE» or press «ENTER».

The block which is loaded remains as a defined block until you choose ***Block Hide*** or else define another block.

Saving part of a layout

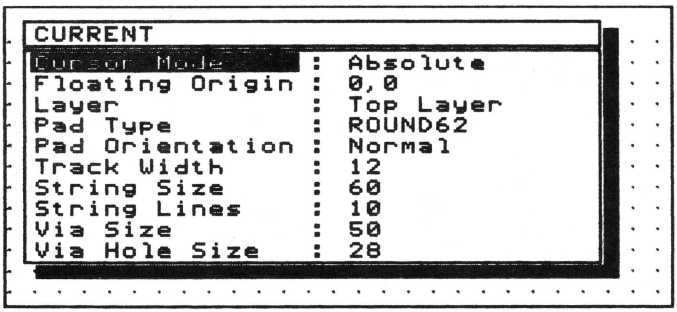
If you identify part of a current layout as a block you might want to use on another occasion, it is a good idea to save it on a disk as a separate file.

To do this, define the section of the layout as a block using the ***Block Define*** option. Then, choose ***Block Write***. You are prompted to supply a 'BLOCK WRITE FILE NAME'.

Enter a filename. If the filename you enter does not exist, the block is written out to that file. If it already exists, you are asked to confirm that you want to overwrite the file. If you do want to overwrite the file choose *Yes*, if not choose *No* then repeat the process with a different filename.

9 VIEWING AND CHANGING THE CURRENT SETTINGS

If you select *Current* from the MAIN MENU you can view the current settings. If you need to change any of the settings then select them from the *Current* menu and enter the new value.



Setting the cursor mode

The cursor mode can either be set to absolute or relative. If the cursor mode is absolute the coordinates measure the X and Y distances from the bottom left hand corner of the board. If the cursor mode is relative then the cursor position is measured relative to some other position on the board. The origin of the relative cursor is set using *Current Floating Origin.*

It is sometimes useful to set a relative cursor position if a number of components need to be placed relative to the position of an important component such as, say, a microprocessor.

Setting a floating origin

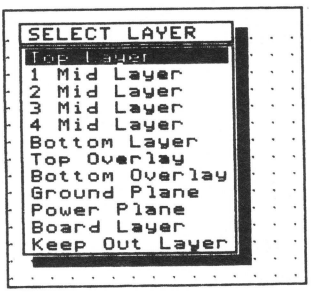
If you want to set the origin (i.e. X: 0 Y: 0) in a position other than the bottom left hand corner of the board, move the cursor to the point which

you want to become the 'floating origin'. Choose *Current Floating Origin*. The origin changes to the current cursor position.

When a floating cursor is set and you move the cursor to the left of the floating origin you will notice that the X value (as shown on the status line) becomes negative. Similarly, if you move the cursor below the floating origin then the Y value becomes negative.

You will not notice any change on the status line unless you set *Current Cursor Mode* to Relative.

The currently selected layer



Current Layer indicates the layer which is currently selected. Choosing the option opens a window which displays all the layers and allows one to become the selected layer.

The selected layer must be turned on before it can be activated. Layers can be turned on and off using *Setup Toggle Layers*.

If you select a layer which is not turned on to be the current layer, the message:

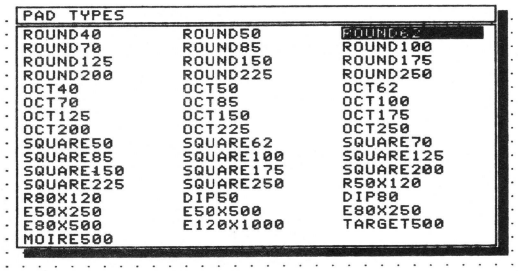
CURRENT LAYER MUST BE TURNED ON

is displayed.

The current pad type

One of the features of *PROTEL-AUTOTRAX* is the ability to define different types of pad. When you choose *Current*, the currently selected pad type is displayed.

To change the currently selected pad type, choose *Current Pad Type*. An information window opens which shows all the available pad types. Move the selection bar to the required type and click «LEFT MOUSE» or press the «ENTER» key.



The default pad type is a circular pad with a diameter of 62 mils and a hole of 28 mils. The default diameter of 62 mils means that if two pads are placed 100 mils apart, then a 12 mil track can be laid between them with a clearance of 13 mils.

Pad orientation

Pads can be placed normally or they can be rotated through 90 degrees. Choose *Normal* or *Rotated* from the *Pad Orientation* menu to select between these options.

This option is very useful since it means that fewer pad types need to be defined.

The default track width

This option allows you to change the default track width. If you do not change it, the default will be set to 12 mils. If you want to change the width, select the option from the menu. An input window will appear, allowing you to type in the new value for the track width.

Track widths can vary from 1 mil to 255 mils.

Checking the text height

The *Current String Size* option allows you to change the height of the text which appears on the board. The height can be changed at any time, but this option should be set to the size which you use most often.

Once again an input window will appear which will allow you to type in the new value. Choose a value from 36 mils to 1000 mils.

Checking the width of string lines

The width of the 'brush strokes' which make up the individual characters in strings can be varied by choosing *Current String Lines*. The normal value is 10 mils, but if you want a particular string to have more emphasis then increase the string width to 15 or 20.

Make sure you choose a width which is compatible with the text height. For example, a width of 10 mils looks reasonable with a text height of 60 mils, but a width of 30 mils makes the text illegible. However, a width of 50 mils looks reasonable with a text height of 500 mils.

The width can vary from 1 mil to 255 mils.

The default via size

The default via size is 50 mils. This can be changed by selecting *Current Via Size* and then entering a new value in the input window.

Vias can vary in size from 1 to 255 mils. Be careful about making vias too large as this will restrict the choices for the automatic router when laying tracks.

The default via hole size

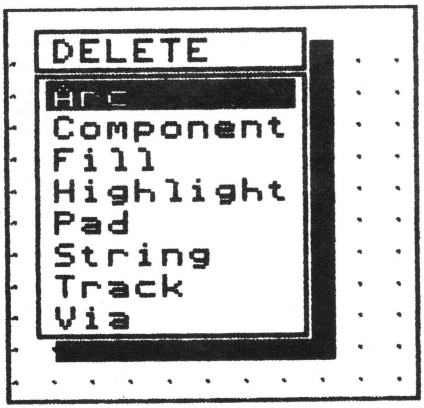
The via hole size is the diameter of the hole through the board. This must obviously be less than the size of the via. The program checks to ensure that a hole size is not specified greater than the via size. If it is larger, the hole size will be changed to a value which is 1 less than the via size.

Thus if the via size is 50 mils and a via hole size of 70 mils is specified the hole size will be automatically changed to 49 mils.

Choose *Current Via Hole Size* to change from the default value of 28 mils.

10 DELETING ITEMS ON THE BOARD

The *Delete* option allows you to delete an arc, component, fill, highlight, pad, string, track or via.



After you delete an item from the board it is often a good idea to refresh the screen by pressing the «END» key. This redraws the screen and eradicates any untidy effects that a deletion may have caused.

When deleting tracks and arcs you need to be able to position the cursor exactly over the item to be deleted. Remember that the cursor is constrained by the snap grid to move in increments of 25 mils. It may be necessary to reduce the size of the snap grid to, say, 15 mils to ensure that the cursor can be positioned correctly.

Deleting an arc

An arc can only be deleted if the currently selected layer (as indicated by the status line and the *Current* menu) is the layer containing the arc. As

they are normally found on the component overlay, this layer should be selected before you attempt the deletion.

Choose *Delete Arc* and you are prompted to select an arc. Move the cursor to anywhere on the arc to be deleted and click the «LEFT MOUSE» button or press «ENTER». You are asked to confirm that you want to delete the highlighted arc. If you choose *Yes* the arc is deleted.

If you hear a beep or if the arc is not deleted you may need to do one of two things. First, check that you are on the correct layer. Second, make sure that the cursor is correctly positioned over the arc. If you find it difficult to position the cursor you might have to change the size of the snap grid.

Deleting a component

You can delete a component regardless of the currently selected layer. There is no need to change to the component layer prior to deleting a component.

To delete a component choose *Delete Component*. You are prompted to select a component. Use the mouse or the arrow keys to move to the component you want to delete.

Now click «LEFT MOUSE» or press «ENTER» to select the component. A window opens asking you to confirm that you really do want to delete the component. Choose *Yes* and the component will be removed from the board.

If you click «LEFT MOUSE» or press «ENTER» when the cursor is not over a component, an input window opens asking you to input the component designator. Enter the component designator of the component you want to delete. Be careful, because you can delete a component that is not currently displayed on the screen!

Deleting a fill

Any fill area you want to delete must be on the currently selected layer.

To delete a fill, simply choose the *Delete Fill* option, move to the required fill and confirm that it is the one you want to delete.

Deleting a highlight

If you want to delete a set of highlighted tracks you must first choose the *Highlight* option from the MAIN MENU. Highlight the required tracks as described in Chapter 14. It is then a good idea to choose *Zoom All* to fit the board on the screen so that you can make sure that it has not highlighted any tracks that you do not intend to be deleted.

Choose *Delete Highlight* and the highlighted section will be deleted. Remember to press «END» to refresh the screen.

If you want to delete an external plane then firstly choose *Highlight Net* to highlight some or all of the plane. Then choose *Delete Highlight* to delete it. You should be aware that an external plane may be broken into many sections and hence you may need to highlight each separately.

Note that deleting a highlight does not delete any components.

Deleting a pad

A pad can be deleted from any layer since it belongs to every layer.

Choose the *Delete Pad* option and the prompt 'SELECT PAD' appears on the status line. Move the cursor to the pad to be deleted and then press «ENTER» or click «LEFT MOUSE». If the pad cannot be deleted you will hear a beep. If it can be deleted a window will appear asking you to confirm that you want to delete the pad - and the pad will appear in inverse. Choose *Yes* and the pad will be deleted.

You will then be asked to select another pad. When you have finished deleting pads press the «ESC» key or click «RIGHT MOUSE».

Do not try to delete a pad which is part of a library component. You need to delete the entire component rather than just one pad.

Deleting a string

To delete a string, you must be on the same layer as the string to be deleted.

Choose the *Delete String* option and the prompt 'SELECT STRING' is displayed. Move to the string you want to delete then press «ENTER» or click the «LEFT MOUSE» button.

The string to be deleted appears in inverse so that you can confirm that it is the one you want to delete. Choose *Yes* to confirm the deletion.

Deleting a track

Note that you must be on the same layer as the track you want to delete.

To delete a track (which is any straight line part of a net) simply choose *Delete Track,* then move the cursor to the track required and either click the «LEFT MOUSE» button or press «ENTER». The track you have chosen appears in inverse and you are be asked to confirm that you want to delete it. Choose *Yes* to delete it or *No* to leave the track there. Press «ESC» or click «RIGHT MOUSE» when you have finished deleting tracks.

If you cannot get the cursor to sit on the track you want to delete change the snap grid size in the *Grid* option of the MAIN MENU to a smaller figure (e.g. from 25 down to 15 mils).

Deleting a via

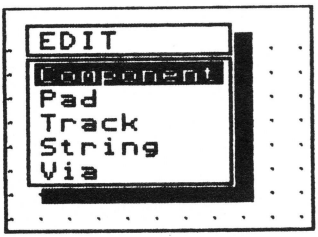
A via can be deleted from any layer.

Choose *Delete Via* then move to the via to be deleted. The prompt 'SELECT VIA' appears on the status line. If your view of the via is not clear enough, press «PGUP» to expand the window.

Click the «LEFT MOUSE» button or press «ENTER» to delete the via. Press «ESC» or click «RIGHT MOUSE» when you have finished deleting vias.

11 CHANGING PART OF THE PCB

The *Edit* menu makes it possible for you to make changes to items that you have already placed on the board.

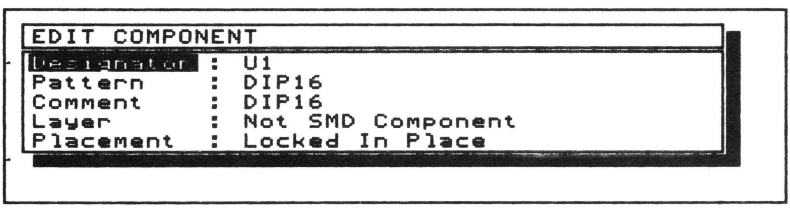


Changing a component

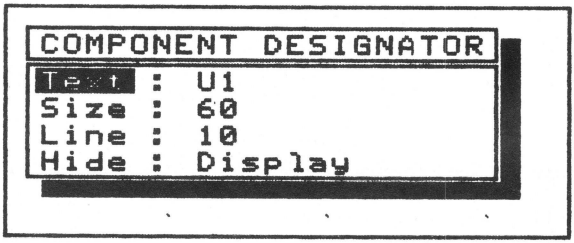
When you choose *Edit Component*, a message appears on the prompt line asking you to select a component. Move the cursor to the component you want to change then click «LEFT MOUSE» or press «ENTER».

If you are working on a tight board and you have overlapped components, a window opens displaying the designators of the components which are overlapping. Select the designator of the component you want to edit.

A window will open showing the designator, pattern, comment, layer and placement status of the selected component.



You can change each of these by moving the selection bar to the required option and pressing «ENTER» or clicking «LEFT MOUSE». If you choose to edit the component designator, a window will open displaying four options.



You can change the text (e.g. change U3 to C4), the size of the text and the width of the string lines. The final option allows you to hide or display the designator. Normally, you would choose to display the designator, but if you are laying down a cluster of components which are the same, you may only want to display the designator and comment for one.

When you have finished editing the designator, click «RIGHT MOUSE» or press «ESC» to return to the *Edit Component* menu.

Changing the *Comment* field is done in the same way as the designator. You should note that it is possible to choose a text height which causes the comment string to overwrite the designator string.

You can change the layer if the component is a Surface Mount Device (SMD). SMDs belong to only one layer. They are glued to that layer - they do not have pins mounted through the board.

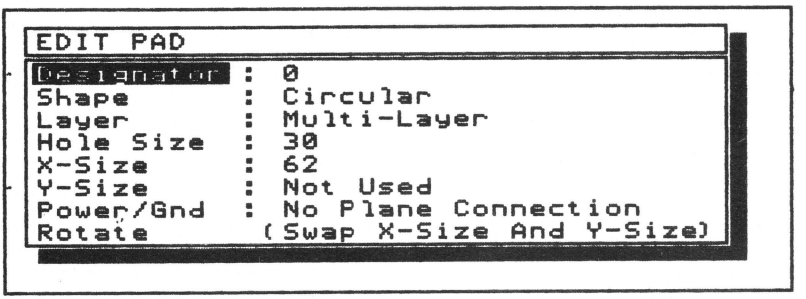
Choose *Edit Component Layer* and a window opens asking you to select either the top layer or bottom layer for the component. If the component is not an SMD, no action will be taken.

If you choose *Edit Component Placement* a window opens giving you the option of choosing 'Free To Move' or 'Locked In Place'. This is important when components are being placed automatically. If you assign 'Locked In Place' to a component, then it will not be moved to

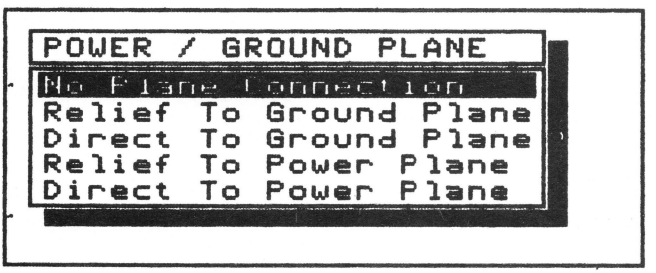
another place on the board. This is important for components such as edge connectors and some critical components such as microprocessor chips.

Changing a pad

If you need to change the size, type or layer of a pad then choose *Edit Pad*. A prompt appears asking you to select the pad. Move the cursor to the pad then press «ENTER» or click «LEFT MOUSE». A window opens showing detailed information about the selected pad.



Most of the features displayed in this window are explained in *Setup Pads* (see Chapter 23). The one additional feature is the ability to change the plane the pad is connected to. Choose *Power/Gnd* and the following window opens.



As you can see, this option can be used to indicate a connection to either the ground plane or the power plane.

Choose the plane that the pad is to be connected to, then press «ENTER» or click «LEFT MOUSE».

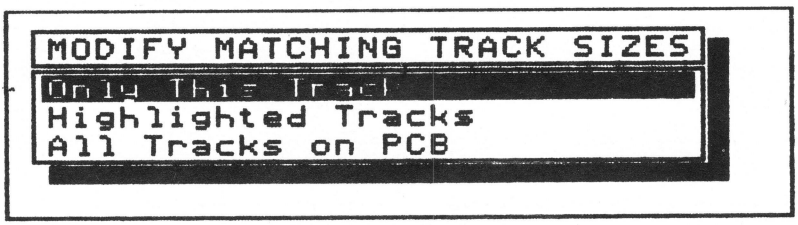
Changing a track

You can change the layer a track is on or you can change the width of a track.

Choose *Edit Track*. A window opens allowing you to change the track layer or the track width.

To change the layer, choose *Layer*. A window opens displaying all the signal and overlay layers which are available. Choose the layer to which you want to transfer the track then press «ENTER» or click «LEFT MOUSE».

To change the track width, choose *Width*, select a track and input the new width (remember that you must be on the same layer as the track you are trying to select). You are asked whether you want to just change the selected track, the highlighted tracks or all tracks on the board.

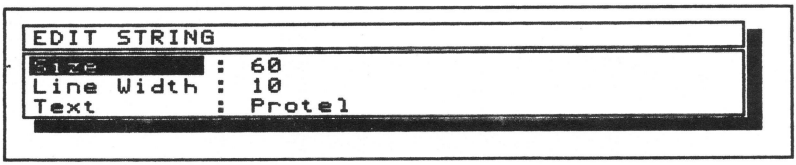


Be careful if you choose the second or third options, global changes to track widths can lead to some unexpected problems.

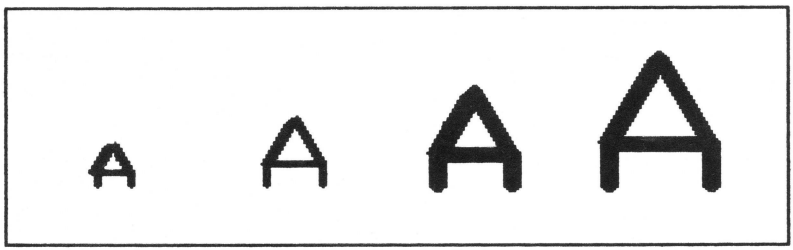
Changing a string

You can change the style of a string or the characters which comprise the string.

Choose *Edit String* and a window opens giving the following options.



Choose *Size* to change the string size which refers to the height of the text. Choose *Line Width* to change the width of the lines which make up the text. As a demonstration of these parameters, the diagram below shows four letter As. Reading from left to right, the letters increase in size. The first two have a line width of 10 mils and the next two have a line width of 20 mils.

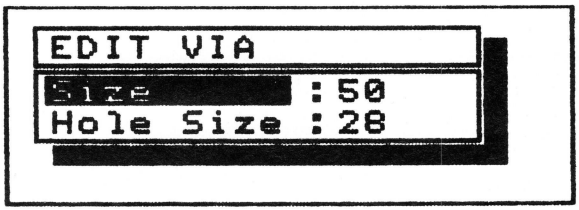


Choose *Text* to change the string text. Remember that all the line editing features described in the Line Editor Tutorial (Chapter 5) are available - there is no need to retype the whole line!

Changing a via

Since vias belong to every layer the only parameters you can change are the via size and the hole size.

Choose *Edit Via*. Select a via by moving the cursor to it, then pressing «ENTER» or clicking the «LEFT MOUSE» button. The following window appears.



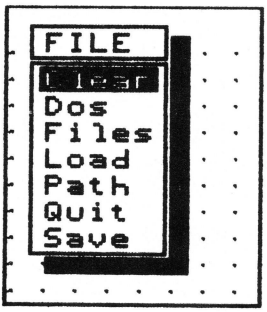
Enter the new via size and hole size (remember that the hole size must be less than the via size) and click «LEFT MOUSE» or press «ENTER». A window opens which gives you the option to modify one via, all highlighted vias or all vias on the board. Select the appropriate option by highlighting it then pressing «ENTER» or clicking «LEFT MOUSE».

Next, you will be prompted to select another via and so on. Finish by clicking «RIGHT MOUSE» or pressing «ESC».

If you try to enter a hole size which is greater than the via size then the hole size will be reduced to one mil less than the via size.

12 MANAGING FILES

The *File* option of the MAIN MENU lets you carry out many of the 'housekeeping' tasks associated with managing files and the work space.



It allows you to save and retrieve files, to set the path to a directory, to inspect a directory for the names of .PCB files and to clear the work space. It also allows you to carry out a range of DOS functions without losing your current work sheet and the associated settings.

Clearing the memory

File Clear allows you to clear the work space when you want to start working on a new board.

Before you invoke this option you must make sure that you have saved all changes to your current work sheet.

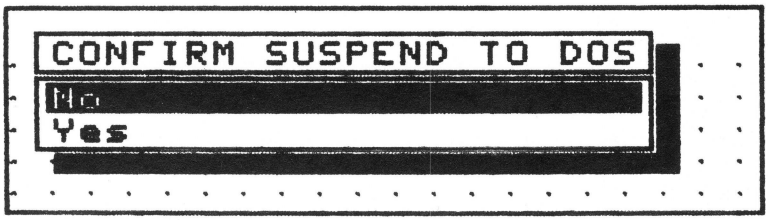
The *Clear* option gives you a chance to change your mind by asking you to confirm the fact that you want to clear the current work space. Choose *Yes* if you want to clear the work space. Choose *No* if you do not want to clear it.

Issuing DOS commands

If you are working in the *PROTEL-AUTOTRAX* program and need to carry out some DOS command that is not available through one of the menus, choose ***File Dos*** to exit the program temporarily.

You are strongly advised to save your current work before invoking this option in case you cannot return to the program for some reason.

You will be asked to confirm the fact that you wish to quit to DOS. If you choose *Yes* the DOS prompt will appear and you can carry out most of the normal DOS commands such as changing drives, inspecting directory lists, copying files and changing directories.



There are no restrictions on the DOS commands that you can issue. One thing that you must remember, however, is not to type **TRAXEDIT** to start the program again as this reloads the program and your previous work may be lost.

Another restriction is that the overlay file called TRAXEDIT.OVR must not be moved to a different directory at this stage.

To return to the program type:

EXIT

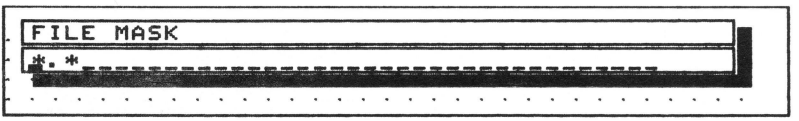
at the DOS prompt and your session will be resumed at the point at which you exited.

Checking files stored on the disk

You will often need to know the names of .PCB files which are already stored on your disc. Choosing ***File Files*** allows you to check the names of all files or of selected files.

When you choose this option an input window opens prompting you to enter a 'mask' for the type of files you want to know about.

The default mask of *.* has been chosen so that all files in the default directory will be listed.



If you enter a more restrictive mask such as *.NET or *.PCB then only files with the extension .NET or .PCB will be listed.

If you enter a specification such as:

CHIP?.PCB

then files with names such as CHIP1.PCB, CHIP2.PCB and CHIPA.PCB might appear.

The selection bar can be moved to highlight a particular filename but no action can be carried out on the file.

Press either mouse button or the «ENTER» key or the «ESC» key to close the window.

Loading a file

This is an option that you will need often. It allows you to load the contents of a *PROTEL-AUTOTRAX* file from the hard disk or the floppy disk into the Random Access Memory of the microcomputer so that you can view or edit it.

Choose *File Load*. You are asked to specify a file name. If you cannot remember the name of the file you can simply click the «LEFT MOUSE» button or press «ENTER» and a list of files in the currently selected directory will be displayed. Move the selection bar to the file you wish to load and click «LEFT MOUSE» or press «ENTER».

A window opens asking you to confirm that this is the file you want to load. If you choose *Yes* then the file is loaded into the memory and the previous contents of the memory are erased. For this reason you should ensure that you have saved your current work before loading another file.

If you want to load a file from a directory which is different to the one set by ***File Path*** then you should type in the file's full path name.

You can also load a .PCB file when you start the *PROTEL-AUTOTRAX* program. If you want to load the file called BOARD1.PCB, then type:

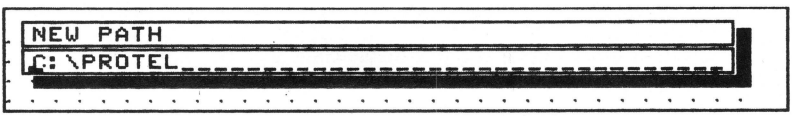
TRAXEDIT BOARD1 and press «ENTER».

The file is then loaded automatically.

Specifying a path name for files

It is a good idea to store your .PCB files in a different directory to the *PROTEL-AUTOTRAX* programs and files.

The path name of the directory in which you are going to store your files can be specified with the ***Path*** option. When you choose ***File Path*** a window opens asking you to specify a new path.



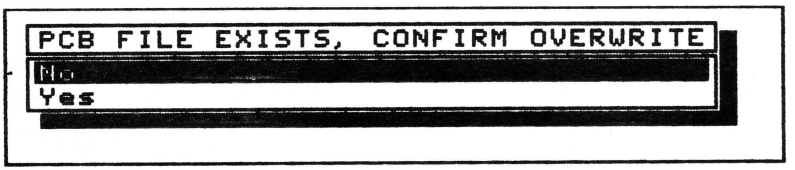
If you have created a directory for your files such as \FILES then that is the path name you should enter. If you have decided that the files should be stored in a subdirectory of the PCB directory then you should enter a path name such as \PCB\FILES. If you want to store your files on a floppy disk then enter a path name such as A:\

Leaving the *PROTEL-AUTOTRAX* program

When you have finished a *PROTEL-AUTOTRAX* session and are ready to leave the program, choose *File Quit*.

You are asked to confirm the fact that you want to quit. If you respond with *No* then you can continue with the session. If you respond with *Yes* then you will be returned to the DOS prompt.

If the board has been changed in any way since the last save operation, another window opens asking you if you want to save the file before you are returned to the DOS prompt. Choose *Yes* to save the file before you exit and *No* to exit without saving the file.



Before turning the computer off remember that it is very important to back up your files by copying them to a floppy disk and then storing the floppy disk in a different, and safe, location.

Saving your files

It is a good idea to save the file you are working on as often as possible. The time between 'saves' will vary depending on the size of the board you are working on.

If you are working on a very large board, it may take a considerable time to save the file and therefore it becomes annoying to save every five minutes. However, you do not want run the risk of having, say, half an hour's work lost when someone trips over your power cord!

Remember that the *PROTEL-AUTOTRAX* program saves your work every thirty minutes unless you change the backup time in *Setup Options Backup*.

When you do decide to save, you will be asked to supply a file name, or else a file name will be specified in the input window. If you want to give the file a new name simply type in the name. If you want to give it the same name as a file which is already stored on the disk then you will be asked to confirm the fact that you want to overwrite the file. Choose *Yes* then click «LEFT MOUSE» or press «ENTER».

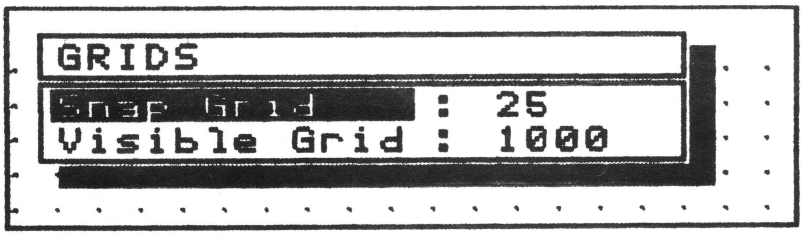
Once the file has been saved then you can continue working.

You should also be aware that when you save a file, a backup of the previous version of the file with the extension .BAK is created. For this reason you must ensure that there is always twice the disk space available than that needed just for the .PCB file.

13 SETTING GRID SIZES

There are two grids which are active in the *PROTEL-AUTOTRAX* program. The first is an invisible 'snap' grid and the second is a visible grid guide which takes the form of an array of dots on the screen.

The distance between points in either grid can be varied by choosing the *Grid* option from the MAIN MENU.



Changing the snap grid

The 'snap' grid is an invisible grid which the cursor follows (or snaps to) whenever the cursor keys or the mouse move the cursor around the screen. As the cursor moves it 'snaps' from one grid position to the next.

Move to the point where you will start connecting the pads, choose *Grid Snap* then change the value to, say, 125. Now each time you move the mouse or press the cursor keys the cursor will snap to the next pad position.

The normal value is 25 mils but this value can be increased or decreased depending on your preference. The current value of the snap grid is always shown on the status line at the bottom of the screen.

One example of where this is useful is when you want to work with edge connector pads which are usually either 125 or 156 mils apart. An example might be if you were going to connect some of the pads of, say,

a STEDGE or S100 connector to the ground or power planes.

When a grid size is selected, the grid points originate at X:0 Y:0 - regardless of whether there is a floating origin. Thus if the cursor is at X:100 Y:100 and you change the snap grid to 30, the cursor would immediately snap to X:90 Y:90 as that is the nearest grid point.

Changing the visible grid

The visible grid helps you to lay components and tracks in a careful and systematic way.

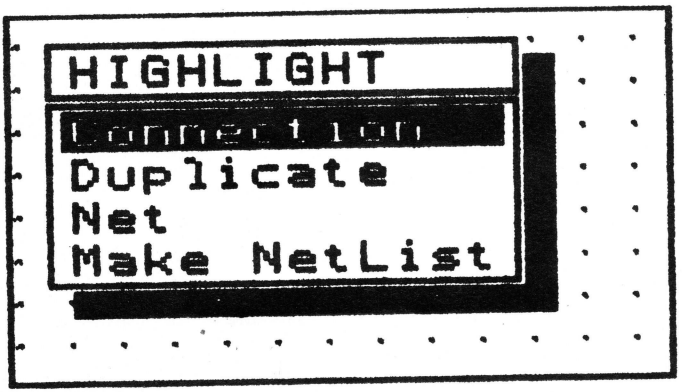
It is usually set to 1000 mils (i.e. one inch) but the value you will need to work with varies greatly depending on the level of magnification you choose. The higher the magnification the lower the value of the visible grid you need.

To change the value, select *Grid Visible Grid* and enter the new value. Press «ENTER» or click «LEFT MOUSE» to exit from the option.

14 HIGHLIGHTING NETS

The *Highlight* option lets you carry out two main functions. First you can highlight a net or a single connection or copy a net and second you can create a netlist based on the existing set of connections.

The highlight option is used widely to verify nets. It helps you make sure that all the connections that need to be made, have been made.



Highlighting a connection

Choose *Highlight Connection* to highlight a connection between two pads. The prompt:

SELECT TRACK/PAD TO HIGHLIGHT

appears on the prompt line. Move the cursor to a track or pad in the connection you want to highlight, then press «ENTER» or click «LEFT MOUSE». The message 'SORTING TRACKS. PLEASE WAIT.' appears at the top of the screen, then the connection is highlighted.

The highlight remains until you either highlight another connection or net or until you select *Highlight Connection* or *Highlight Net* and

highlight a blank area of the board.

Duplicating nets

The ***Highlight Duplicate*** option provides you with a simple means of duplicating nets such as memory routes.

To duplicate a net choose ***Highlight Duplicate*** and the message:

SELECT COPY SOURCE

appears on the prompt line. Select the point on the highlighted net that you want to act as the reference point then press «ENTER» or click «LEFT MOUSE». Now the message:

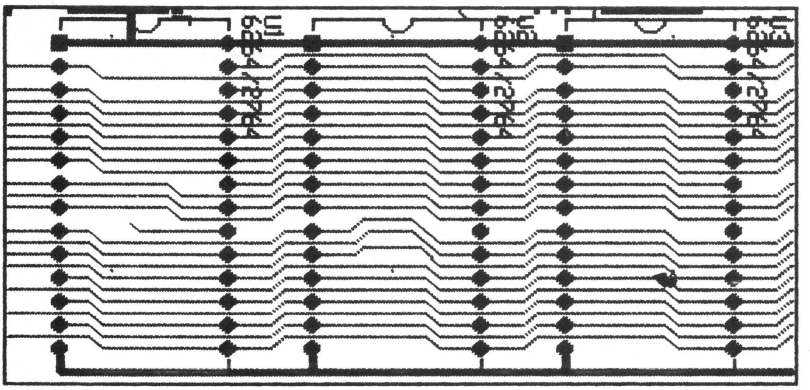
SELECT COPY DESTINATION

appears on the prompt line. Move the cursor to the point you want to copy the net to and press «ENTER» or click «LEFT MOUSE». The net is duplicated and the prompt line indicates that you can copy the net again. You can continue making duplicate copies of the net until you press «ESC» or click «RIGHT MOUSE».

If no net is highlighted nothing is duplicated.

An example of the use of the ***Highlight Duplicate*** option is the duplication of memory routes. A memory route is usually a short pad to pad connection which is only on one layer and which may comprise two or three short track segments. It is not possible to duplicate such a pattern using the ***Repeat*** option as that only repeats the placing of the last track segment.

The example shown in the diagram below is taken from the DEMO board supplied with *AUTOTRAX*. It shows some of the memory routes. You can see the regular pattern which could be duplicated using this option.



Highlighting a net

The *Highlight Net* option is similar to the *Highlight Connection* option. The only difference is that the *Highlight Net* option highlights a complete net, whereas *Highlight Connection* only highlights one connection.

Creating a netlist

If you have routed a board and want to create a netlist, use the *Highlight Make Netlist* option.

One reason why you might want to produce a netlist after you have made all the connections is to obtain a printed copy of the netlist for manual checking of connections. Another reason might be that you do not have the *PROTEL-SCHEMATIC* program which produces netlists and have had to make all connections using the pad to pad router. You might want to create a netlist and submit the netlist to the automatic router to produce a better result.

When you choose *Highlight Make Netlist*, an input window with the heading 'NETLIST FILENAME' opens. At this stage you should enter the name for the netlist file which is to be created.

You are now asked to confirm that you want the netlist generation to proceed. Choose *Yes* and the nets are highlighted one by one as they are

stored in the file. You will notice that on the right hand side of the status line a **recursion depth count** is provided. This simply indicates how hard the program is 'thinking' in working out the nets!

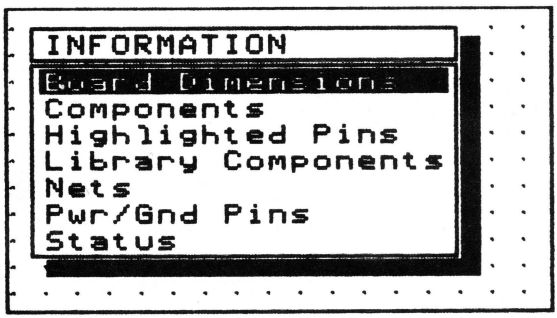
The netlist generation continues until all nets have been stored in the file. At this stage the status line is restored and the cursor reappears on the board.

The file that is saved has the extension .NET.

15 GETTING INFORMATION ABOUT THE BOARD

The *Information* menu is extremely useful in providing information about the state of the board on which you are currently working.

Essential facts such as the number of arcs, components, pads, holes and vias as well as the amount of memory which is available can be determined with this option.

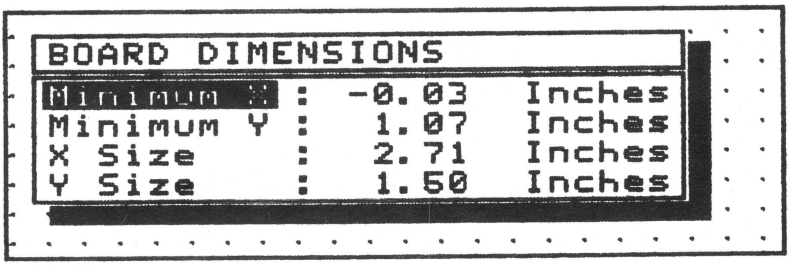


Determining the board's dimensions

Choose *Information Board Dimensions* to view an information window which displays the current size of the board.

The minimum and maximum values of the X and Y coordinates are displayed. It is possible that the minimum values can be negative. The reason is that if an item such as a pad is placed at, say, X: 0 Y: 500, the left hand side of the pad will spill over into the negative area.

If the diameter of the pad is 62 mils, then you will have extended the board in the negative direction by 31 mils.



Listing the components already placed

To obtain a list of all the components you have already placed on the board choose *Information Components*. *PROTEL-AUTOTRAX* scans the components already placed and lists them in an information window.

Checking highlighted pins

If you highlight a connection or a net using *Highlight Net* or *Highlight Connection* then *Information Highlighted Pins* can be used to provide you with a list of the pins in that net. This is useful if you want to check to make sure you have connected all the pins in a particular net.

If you choose *Information Highlighted Pins* when no pins have been highlighted, the message:

NO PINS HAVE BEEN HIGHLIGHTED

is displayed.

Press «ENTER», «ESC» or click either mouse button to exit this window.

Checking the names of library components

To find the names of components stored in the currently selected library, choose *Information Library Components*.

A window opens showing the names of all components in the current library. Use «PGDN» to move to the next screen if there are too many names.

Press «ENTER», «ESC» or click either mouse button to exit this window.

Listing all the nets

This option allows you to obtain a list of all nets which are currently in the memory. Choose *Information Nets* and a window listing the nets opens. Remember that if there is more than one window-full, press «PGDN» to see more.

Press «ENTER», «ESC» or click either mouse button to exit this window.

If no nets have been loaded with the *Netlist Get Nets* option then the message:

NO NETS LOADED

is displayed.

Listing power and ground plane nets

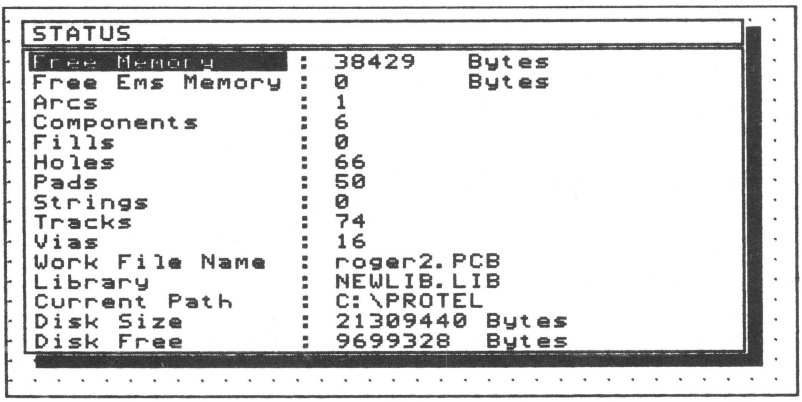
To obtain a list of all the power nets or ground nets, choose *Information Pwr/Gnd Pins*. A menu appears which allows you to choose the ground plane or the power plane.

Choose *Ground Plane* to list all connections to the ground plane. Choose *Power Plane* to list all connections to the power plane.

Press «ENTER», «ESC» or click either mouse button to exit this window.

Essential information about the board

This option allows you to check information about the board. Choose *Information Status*. An information window opens which shows the following details.



The display shows the number of bytes of free memory and expanded (EMS) memory. When you are creating a large board and are worried about the amount of free memory, it is a good idea to check this status screen from time to time.

It also displays the number of arcs, components, fills, holes, pads, strings, tracks and vias on the board as well as the board's filename and the name of the current library.

This information can be quite useful. If you are trying to minimize the number of vias, for example, you can check this screen to see how many are currently on the board.

The bottom of this window shows the currently active path as well as the disk size and the number of free bytes of memory available on the disk.

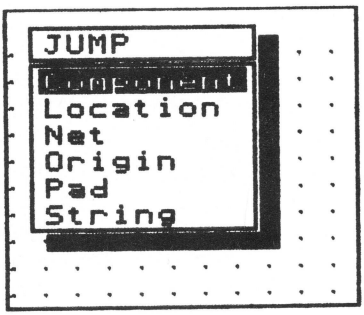
Press «ENTER», «ESC» or click either mouse button to exit this window.

16 USING THE JUMP OPTION

The *Jump* option is particularly useful on large boards when you need to jump to a specific component, locate a string or pad, or move to a particular location.

Remember that the position of the cursor on the board is defined by its distance in mils from the origin in both the X and Y directions. Looking at the screen, the X direction is horizontally towards the right and the Y direction is vertically upwards.

This may be different, however, if the origin has been changed in the *Current* menu.



Jumping to a component

You will often find it necessary to jump to a specified component on the board.

Choose *Jump Component*. An input window opens asking for the component designator.

If you can't remember the designator, simply click «LEFT MOUSE» or press «ENTER» and an information window opens showing all component designators on the board.

Move the selection bar to the designator required and click «LEFT MOUSE» or press «ENTER». The cursor jumps to pin 1 of the component you requested.

Try this with the DEMO.PCB file loaded. Choose *Jump Component*, then input U5 which is the designator for the Z80 microprocessor. Notice that the cursor jumps to the square pad which specifies pin 1 on the chip.

Jumping to a location

This option allows you to specify a location on the board for the cursor to jump to. Choose *Jump Location*. A window opens asking you to input the X location. The value in the window shows the current position of the cursor. Type in the new value. A window now opens asking for the Y location. Type in the Y value and press «ENTER» or click «LEFT MOUSE».

The cursor immediately relocates at the new coordinate.

It may appear to be strange that the default value given in the input window is the current cursor position. If you are already at that position why would you want to jump there? The reason is that in specifying a coordinate to jump to, you may make a mistake in typing the numbers. If you do then you can easily edit the incorrect number rather than typing both coordinates in again.

Jumping to a net

This option allows you to position the cursor on a particular net.

Choose *Jump Net*. An input window opens asking you to enter the name of the net. If you cannot remember the name enter a «?» and an information window listing all the net names opens. You can then select the appropriate net name by moving the selection bar to the name and pressing «ENTER» or clicking «LEFT MOUSE».

If the net exists then the cursor jumps to the nearest pad in that net.

If the net does not exist or if no nets are loaded then the message:

NET NAME NOT FOUND

or the message:

NO NETS LOADED

is displayed.

With the DEMO.PCB file and the DEMO.NET file loaded try jumping to the net CPUCLK and the net GND.

Jumping to the origin

This option simply relocates the cursor at the absolute origin in the bottom left hand corner of the board, regardless of whether a floating origin has been set.

Choose *Jump Origin* and the cursor is immediately relocated.

Jumping to a pad

This option allows you to specify a component to jump to. Choose *Jump Pad*. Select the component by jumping to the component or by moving to it with the mouse or the arrow keys. Press «ENTER» or click «LEFT MOUSE».

An input window then opens asking you to enter a pin number. Type in a pin number and the cursor jumps to that pad.

This is useful when connecting pads if you need to remember the order in which the pins are numbered. This is not usually a problem with ICs, but it is possible to forget the numbering system for connectors such as DB25s.

If the pin number that you specify does not exist (e.g. if you input pin 17 for a DIP16 pattern) then the message:

PAD NOT FOUND

is displayed.

Jumping to a string

This option allows you to jump to a string that you specify. Choose *Jump String*. An input window opens asking you to enter the string to jump to.

If there is more than one matching string on the board *PROTEL-AUTOTRAX* will do an exact match first. Therefore if you type in **Pro**, it will firstly find the string **Pro**, then **PRO** then **pro** then any other beginning with **Pro...** and so on.

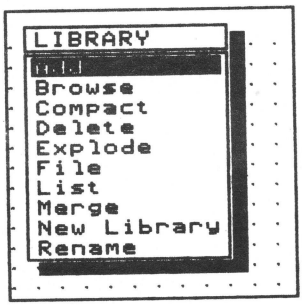
If you attempt to jump to a string which does not exist, the message:

STRING NOT FOUND

is displayed.

17 MANAGING LIBRARIES

The *Library* menu provides the options you need to create new library components, manage existing libraries of components and create new libraries.



Adding components to the library

Before you can add a component to the library you must first create it with the normal tools available through the *PROTEL-AUTOTRAX* program. Most components can be created with tracks, arcs and pads. The procedure for doing this is described in detail in Chapter 4.

When you have created the component, mark it as a block using the *Block Define* option.

Then choose *Library Add.* Enter a name for the new component. The message:

CREATING LIBRARY COMPONENT

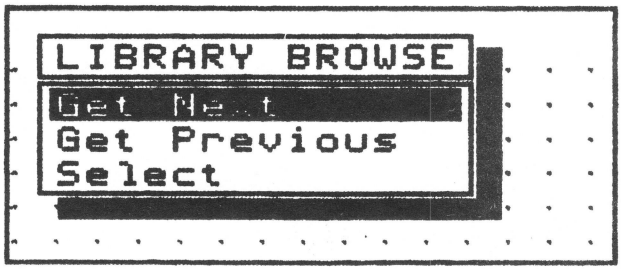
is displayed briefly as the component is added to the currently selected library.

Browsing through library components

The *Library Browse* option allows you to inspect the patterns of

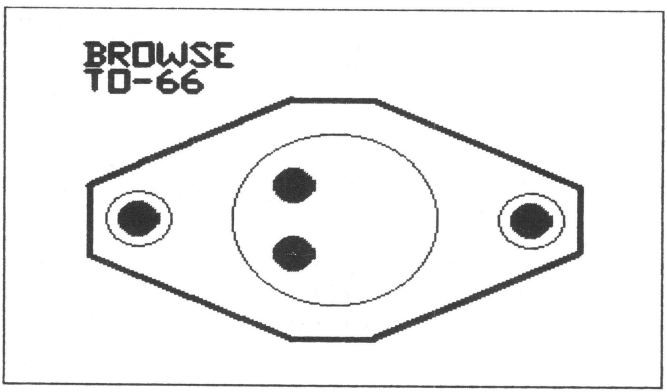
components stored in the library. You can either search backwards or forwards through the library or else inspect a particular pattern.

Choose *Library Browse*. The screen is cleared (don't worry you haven't lost your work!) and the following menu appears.



Choose *Library Browse Get Next* to display the next component in the library. You may need to expand the screen using «PGUP» to be able to see it properly. While you are in the browse option you can use the «PGUP», «PGDN» and cursor keys.

When the pattern of the component is displayed, the prompt line contains the message 'LIBRARY BROWSE'. This is to remind you that to return to the menu you should press «ENTER» or click «LEFT MOUSE».



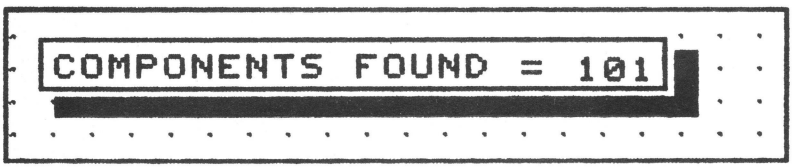
Choose *Library Browse Select* to display a particular component. An input window opens prompting you to enter the name of a component. If you cannot remember the name enter a «?» and an information window listing all the names opens. Select the component by moving the selection bar to the correct name then clicking «LEFT MOUSE» or pressing «ENTER».

Press «ESC» or click «RIGHT MOUSE» to exit from this option. When you do, the board you are working on reappears.

Compacting the library

If you add and delete a number of components, then it is possible that the file will become fragmented. This may be a problem as it will take up more space than necessary on the disk.

Choose *Library Compact* and a window opens asking you to confirm that you want to compact the library. If you choose *Yes*, another window opens which lists each of the components found during the compacting process. When the process is complete a message similar to the following appears.



Press «ENTER» or click «LEFT MOUSE» to finish the compacting.

Deleting a component from the library

To delete a component from the library, choose *Library Delete* and enter the name of the component you want to delete. If you do not know the name of the component, simply press «ENTER» or click the «LEFT MOUSE» button when you see the ? in the input window. Select the component you want to delete.

You are asked to confirm the fact that you want to delete the component. Choose *Yes* to delete it.

'Exploding' a component

Often, when you want to design a new component there is already a component in the library which is similar and may only need minor modifications. If that is the case you can place the existing library component on the board and explode it. Having 'exploded' the component, you can edit it and then save the edited version as a new library component.

Choose *Library Explode* and you are asked to select the component you want to 'explode'. Move to the component and press «ENTER» or click «LEFT MOUSE».

The component will appear to be basically unaltered (except for the designator and comment which will be removed) but each element of the component will now be in its primitive state. That is, instead of a single component, it is now a collection of tracks, arcs, and pads.

To convince yourself of this, move to any other component and select the top overlay (by pressing «+» or «-» on the keypad). Now choose *Edit Track* and move the cursor over the track portion of a component. You will not be allowed to edit the track! Try the same thing with a component which you have exploded - you can now change the track.

Once you have changed the pattern you must then define it as a block and add it to the library as a new component as described in the tutorial in Chapter 4.

This option is also useful if you want to make minor adjustments to existing library patterns.

Choosing a different library file

Choose the *Library File* option to change the currently selected library file. A different directory or disk drive can be specified if required.

A window opens with the prompt 'LIBRARY FILE NAME'.

Enter the name of the library file you want to make active. If you cannot remember the name, enter a «?» and a list of the .LIB files is displayed.

Move the selection bar to the one you want to open then press «ENTER» or click «LEFT MOUSE».

Remember that you can check the name of the currently active library by choosing *Information Status.*

Listing the library components

Choose *Library List* to list the names of components in the currently selected library. The message 'WORKING. PLEASE WAIT.' is displayed at the top of the screen and then an information window will open displaying the names of the components in the library.

If you need to check the names of components in other libraries you must first change the currently selected library with *Library Files* as described above.

Merging libraries

This option allows you to move a component from another library into the existing library.

Choose *Library Merge.* You are asked to enter the name of the library to merge **from.** Enter the name of the library **from** which you want to move the component. You are then asked to enter the name of the component you want to merge. If the component exists then it will be added to the currently selected library.

Creating a new library

This option allows you to create a new library. Choose *Library New Library.* You are prompted to supply the name for the new library. A new library file is created on the disk with the extension .LIB.

If you try to specify a name for the new library which is the same as a library which already exists, then the message:

LIBRARY FILE ALREADY EXISTS

will be displayed.

It removes the existing library from the memory (but not from the disk) and allows you to create a library with a new set of components. You

can, of course, use *Library Merge* to move components from other libraries into the new library.

Renaming a component

This option allows you to rename a component. Choose *Library Rename*. You are asked to rename a component in the currently selected library. The message:

NAME IN LIBRARY

appears.

Enter the name of the component whose name you want to change. The message:

NEW LIBRARY COMPONENT NAME

then appears. Enter the new name for the component.

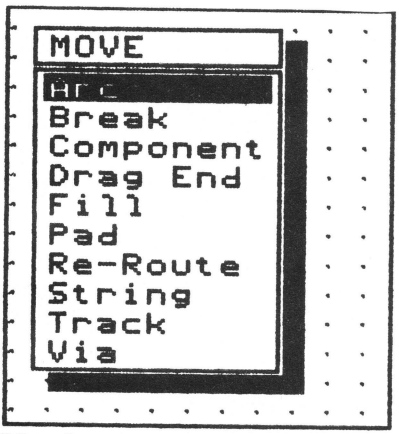
If the component does not exist, the message:

COMPONENT NOT FOUND IN LIBRARY

is displayed.

18 MOVING ITEMS ON THE BOARD

This is a particularly important option for 'fine tuning' a board layout. Items such as pads, vias, components, fills, arcs and strings can be moved to another place on the board. But it is the flexibility for controlling tracks that you will find most useful. You can move a track, break a track at a particular place, drag a track from one of its endpoints or re-route a track. You will also find it useful for adjusting the placement of components after they have been placed with the auto place routine.



Moving an arc

Choose *Move Arc* to move an arc. The message 'SELECT ARC' appears on the prompt line. Move the cursor to the arc you are going to move then press «ENTER» or click «LEFT MOUSE».

Use the mouse or the cursor keys to move the outline of the arc to its new position. Fix the position by pressing «ENTER» or clicking «LEFT MOUSE».

You are now prompted to select another arc. If you have finished moving arcs, press «ESC» or click «RIGHT MOUSE». Otherwise you can select another arc and move it in the same way.

Breaking a track

Sometimes you may want to make a track shorter or break a track into two sections. To do this choose *Move Break*. The prompt 'SELECT TRACK' appears. Move the cursor to the exact spot on the track where you want to break it.

The track now acts like a 'rubber band'. If you move the cursor, the track stretches in two sections as if it were a rubber band held at the cursor position. The message:

DRAGGING BROKEN TRACK

is displayed on the prompt line.

When you have placed the tracks, press «ENTER» or click «LEFT MOUSE».

The important thing to note about this option is that it 'breaks' the track into two tracks. Each part will now be recognised as a track so one section can be deleted if required. This is a simple way of making a track shorter.

Moving a component

Choose *Move Component* to move a component to a new position on the board.

The message 'SELECT COMPONENT' on the prompt line indicates that you should move the cursor to the component that you want to move. Press «ENTER» or click «LEFT MOUSE» to select the component.

The message on the prompt line will be replaced with 'MOVING COMPONENT'. As you move the cursor, an outline of the component also moves.

Place the component in its new position by moving it with the mouse or the cursor keys. Refer to page 66 for details of how to rotate the component.

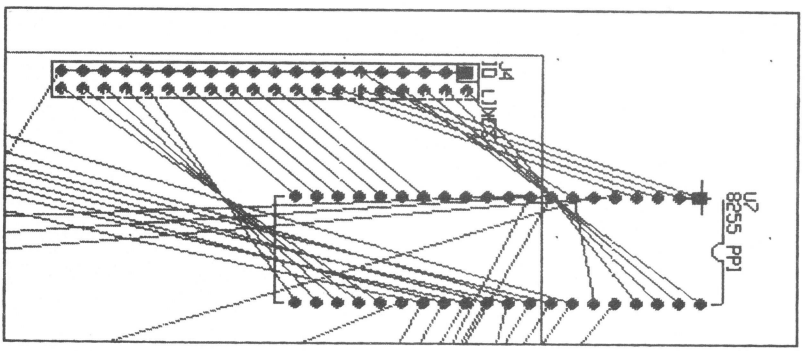
A window opens asking if you want to 'DRAG TRACKS'.

If you then want the tracks to retain their 'connectivity' then select *Yes*. At this stage all tracks are stretched to reach the new position of the component. Don't worry about the fact that the tracks are distorted. You can re-route them later. If you choose *No*, the component is moved but the tracks will remain in position, thus breaking any connections.

A powerful feature of the *Move Component* option is that it can be used to 'fine tune' the placement of components prior to routing. To understand the process, work through the following example.

Choose *File Load* and load the unrouted PCB called PLACED. Now choose *Netlist Get Nets* and load the netlist called DEMO. Now choose *Netlist Show All Nets*. You will now see the 'ratsnest' of connections displayed.

Now choose *Move Component* and select the component with the designator U7. Move the component and notice how the ratsnest moves with the component. Using this technique you can move critical components to other places on the board to try to reduce track congestion.



Dragging the end of a track

This option allows you to drag the end of a track to another place on the board while maintaining its connection to other tracks.

Choose *Move Drag Track* and you are prompted to select a track. Move to the track you want to drag and click «LEFT MOUSE» or press «ENTER». The cursor jumps to the closest track end and the message:

DRAGGING TRACK END

appears on the prompt line. You can now drag the track to the new position. Any connected track will also be dragged and vias will be repositioned.

Note that the track you select must be on the currently selected layer.

Moving a fill

Choose *Move Fill* to move a fill. The message 'SELECT FILL' appears on the prompt line. Move the cursor to the fill you are going to move then press «ENTER» or click «LEFT MOUSE».

The message 'MOVING FILL' on the prompt line indicates that you should use the mouse or the cursor keys to move the outline of the fill to its new position. Fix the position by pressing «ENTER» or clicking «LEFT MOUSE».

You will now be prompted to select another fill. If you have finished moving them, press «ESC» or click «RIGHT MOUSE».

Moving a pad

Choose *Move Pad* if you want to move a pad. The message 'SELECT PAD' appears on the prompt line. Move the cursor to the pad you are going to move then press «ENTER» or click «LEFT MOUSE».

Use the mouse or the cursor keys to move the outline of the pad to its new position. Fix the position by pressing «ENTER» or clicking «LEFT MOUSE».

You are now prompted to select another pad. If you have finished moving pads, press «ESC» or click «RIGHT MOUSE». Otherwise you can select another pad and move it in the same way.

Re-routing a track

The *Move Re-Route* option allows you to manually route a connection by breaking the track at a number of points. The sections remain anchored to the pads or vias at either end.

You break the tracks and choose the new route in the same way as that described in the *Move Break* section above.

Moving a string

Choose *Move String* if you want to move a string. The message 'SELECT STRING' appears on the prompt line. Move the cursor to the string you are going to move then press «ENTER» or click «LEFT MOUSE». Note that you can position the cursor anywhere on the string.

Use the mouse or the cursor keys to move the outline of the string to its new position. Refer to page 66 for details of how to rotate the string. Fix the position by pressing «ENTER» or clicking «LEFT MOUSE».

You are now prompted to select another string. If you have finished moving strings, press «ESC» or click «RIGHT MOUSE». Otherwise you can select another string and move it in the same way.

Moving a track

When you want to move a track, choose *Move Track* and then move the cursor to the track you are going to move. Remember that a track is a single line segment, not the entire connection between two pads.

Select the track you want to move by pressing «ENTER» or clicking «LEFT MOUSE». The message 'MOVING TRACK' appears on the prompt line. At this stage you can move the track to the required position.

When you move the track, connectivity with vias and related tracks is maintained, so connected tracks and vias are dragged with the track. Pads are not moved with the track, however.

Moving a via

Choose *Move Via* if you want to move a via. The message 'SELECT VIA' appears on the prompt line. Move the cursor to the via you are going to move then press «ENTER» or click «LEFT MOUSE».

Use the mouse or the cursor keys to move the outline of the via to its new position. Fix the position by pressing «ENTER» or clicking «LEFT MOUSE».

You are now prompted to select another via. If you have finished moving vias, press «ESC» or click «RIGHT MOUSE». Otherwise you can select another via and move it in the same way.

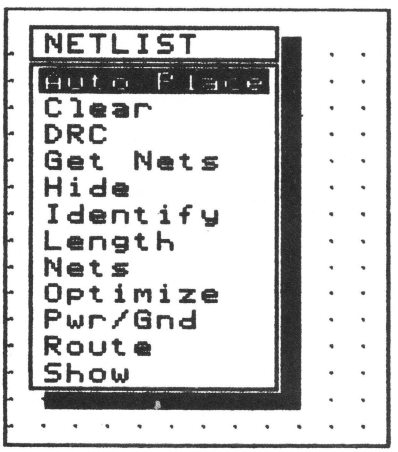
Note that the tracks which are connected to the via are stretched to maintain their connectivity.

19 MANAGING NETLISTS

The most impressive feature of *PROTEL-AUTOTRAX* is its ability to automatically route a board based on a netlist supplied from the *PROTEL-SCHEMATIC* program or generated from the *PROTEL-AUTOTRAX* package itself. Another important feature is the ability to place components from the information in a netlist. This also saves a lot of the tedious work associated with placing components on a board.

The routing algorithm is highly efficient and a high percentage completion is achieved. The time taken to route a board is also short compared with traditional approaches.

Before you can route a board there are a number of 'housekeeping' functions related to netlists which you need to be aware of. The *Netlist* menu provides the facilities you require.



You will notice that the *Route* option is part of the *Netlist* menu. Due to the importance of this option it is treated separately in the next chapter.

Placing components automatically

Once you have generated a netlist with the *PROTEL-SCHEMATIC* package, you can have the components placed automatically by choosing the *Netlist Auto Place* option.

This option has been designed to reduce the tedium involved in placing many components on a board. It does not necessarily produce the best placement and some 'tidying' work inevitably needs to be done. As a rough estimate it may reduce the time you spend placing components by up to 90%.

Before you begin it is a good idea to follow two simple rules. The first is to manually place any edge connectors or other components which must be in a particular place on the board. Placing them manually automatically marks the component as 'Locked in Place'. The second rule is to place any 'seed' components which you want to build the board around. An example of a seed component might be a microprocessor chip.

The board will be 'built' out from the corners of the board, from the corners of keep out areas and from the corners of pre-placed components.

Be aware that with very small boards some strange things may happen. Nevertheless it is still quicker to use the *Auto Place* routine then move the components manually.

If the message:

NO OUTLINE ON THE KEEPOUT LAYER

is displayed this means that you have not defined the border within which the components are to be placed. To overcome this, turn the Keepout layer 'on' with *Setup Toggle Layers*, then select that layer by pressing the «+» key on the numeric keypad.

Draw an outline using *Place Track* which defines the limits of the board. Now hide the Keepout layer again and return to the *Auto Place* routine.

Clearing a netlist

If you want to load a netlist you sometimes need to clear an existing netlist from the memory. Also when you have finished routing, you may want to clear the netlist to reduce the amount of memory used.

Choose *Netlist Clear*. You are asked to confirm that you want to clear the nets from the memory. Choose *Yes* to confirm the action.

If you choose this option when there is no netlist in the memory the message:

NO NETS LOADED

is displayed.

Checking the design rules

The *Netlist DRC* option allows you to check to see that the tracks, vias and pads have been placed according to a set of rules which you have nominated. The process is known as **design rule checking**.

The process is particularly important after you have laid some tracks manually.

The *DRC* option first loads the netlist file, then checks to see that all nodes in each net are connected. If they are not, an error is generated and the net is broken into sub-nets. At this stage it also checks to see if there are any extra pins.

In the next step all nodes are highlighted and a check is made to see if anything which is not highlighted is closer than the design rule check allows.

Loading a netlist

One simple way of creating a netlist that contains the information *PROTEL-AUTOTRAX* requires to lay all the necessary tracks is to use the *PROTEL-SCHEMATIC* package. You can also generate a netlist using the *Make Netlist* option in the *Highlight* menu.

In each case the netlist is stored on the disk in a file with a .NET extension.

To load a netlist choose *Netlist Get Nets*. An input window opens prompting you to enter the filename of the netlist you want to load. The window displays the default value *.NET.

If you cannot remember the filename simply press «ENTER» or click the «LEFT MOUSE» button.

A window opens displaying the names of the files with a .NET extension. Move the selection bar to the file you want to load and click «LEFT MOUSE» or press «ENTER».

Each net is now loaded and the names are displayed in a window at the top of the screen. You are then asked to select the optimization method you want to use. Select one of the methods and the nets are optimized to reduce the overall length of each net. You are then returned to the *Netlist* menu.

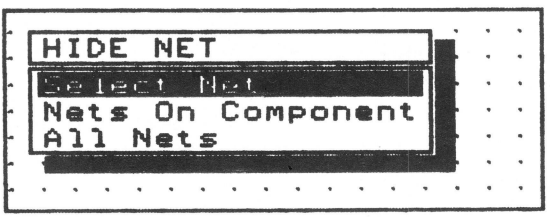
If you try to load a netlist when there is already one in the memory, the message:

NETLIST ALREADY LOADED, CONFIRM LOAD

will be displayed. Choose *Yes* if you want to load the new netlist and overwrite the one in the memory.

Hiding nets

If you have already displayed a set of nets (perhaps through the *Show* option) you can hide some or all of them with the *Netlist Hide* option. The following window appears.



Choose *Select Nets* to specify the names of some of the nets, or choose *All Nets* to hide all the nets which are currently displayed.

Identifying nets

Once you have loaded a netlist, you can identify any net with the *Netlist Identify* option. Firstly, you need to choose *Netlist Show All Nets* so that you can see the 'ratsnest' of connections.

When you choose *Identify* you are prompted to select a connection. Move the cursor to the net you want to identify then press «ENTER» or click the «LEFT MOUSE» button.

A window now appears displaying the net name of the connection pointed to by the cursor. Press «ESC» or click «RIGHT MOUSE» to exit from this option.

Determining the connection length

If you are concerned about the total length of the connections and, perhaps, want to optimize the them, then you can determine the total length by choosing *Netlist Length.* A window opens showing the total connection length in inches.

Listing net names

Choose *Netlist Nets* to obtain a list of the names of all nets currently loaded. Remember that you can use the «PGUP» and «PGDN» keys to scroll through the names if there are more than will fit in one window.

Optimizing nets

Nets are automatically optimized when they are loaded from the disk. Each net is checked to determine the shortest path between the various nodes. If you have changed a number of nets you may need to optimize again before you route the board.

To do this choose *Netlist Optimize*. You are asked to select the nets you want to optimize. Now choose *Yes* to confirm the fact that you want to optimize the nets. Then specify an X-bias or a Y-bias when you choose to optimize. This allows you to make sure that decoupling capacitors and similar components are not connected in an 'unpleasing' way.

Specifying power and ground plane nets

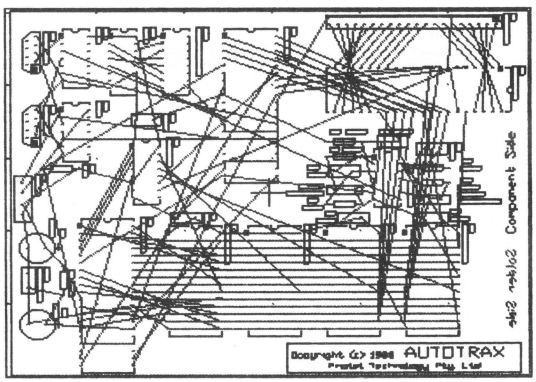
Choose *Netlist Pwr/Gnd* to specify the nets which you want to specify for the *Ground Plane* and the *Power Plane*.

Routing a board

Netlist Route is such an important part of the *PROTEL-AUTOTRAX* program that it is described in the next chapter entitled 'Routing a board'.

Showing highlights

Choose *Netlist Show* to highlight selected nets or all nets. If the *All Nets* option is chosen, a complete highlight of all connections will be displayed in white. This is known as a 'ratsnest'.

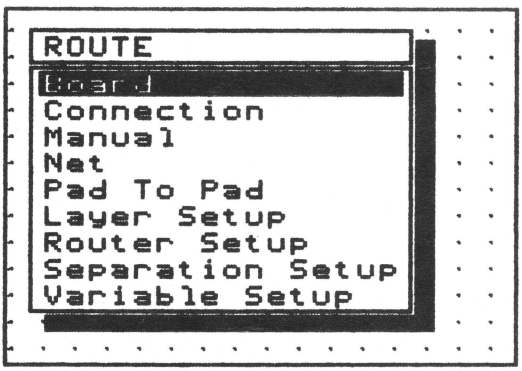


If you only want to highlight a few nets then choose *Select Nets*. You will be asked to specify the name of the net. When you do, the net will be highlighted. It is a good idea to choose *Zoom All* before doing this so that you can see the complete net.

20 ROUTING A BOARD

This powerful feature of the *PROTEL-AUTOTRAX* package will save you many hours of work. It allows you to route a board either manually using the pad to pad router or automatically from a netlist generated by *PROTEL-SCHEMATIC* or *PROTEL-AUTOTRAX*.

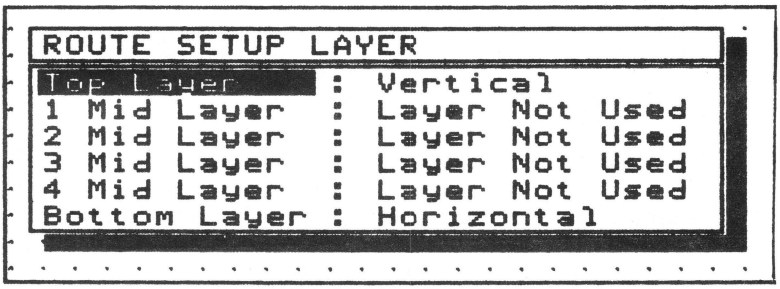
In this section the options which are described do not strictly follow the order in which they appear in the ***Route*** menu. The setup options have been described first.



A short tutorial has been included in the *Netlist Route Board* section for those who are not familiar with the process. Remember that the tutorial entitled *Laying out a PCB* also has some hints for using the router.

Layer setup

The normal practice in laying tracks on a PCB is to place them vertically on one layer and then horizontally on the next and so on. When you choose *Netlist Route Layer Setup* you are given the opportunity to choose horizontal tracks, vertical tracks or no tracks for each of the signal layers.


ROUTE SETUP LAYER
Top Layer : Vertical
1 Mid Layer : Layer Not Used
2 Mid Layer : Layer Not Used
3 Mid Layer : Layer Not Used
4 Mid Layer : Layer Not Used
Bottom Layer : Horizontal

Choosing either *Vertical* or *Horizontal* does not constrain the router to produce exactly vertical or horizontal connections. Often a short 45 degree track will be placed so that a pad or via can be avoided.

Setting up the router passes

When you choose *Netlist Route Route Setup* you are presented with a list of 'passes' which the router can make when it is routing the board (or a net).

You can choose to activate or deactivate any of these passes. Before deciding, however it is a good idea to be aware of the function of each of them. These are now described.

Pre-Routes

The default status for this pass is 'off', since many people do not lay any tracks prior to routing. If you do have a particular reason for laying some tracks manually prior to routing, then you must turn this option on.

If you don't pre-route a partially completed board, the router will not take the existing tracks into account when it chooses the path for new tracks. This may lead to tracks overlapping on the same layer.

When the pre-route is 'on' the first pass checks all existing connections and notes the position of all tracks on the board.

Memory Routes

The memory route pass places short horizontal or vertical tracks. This route is particularly useful for routing the tracks which join the pads of memory chips.

Horizontal

During this pass, horizontal tracks (with some short 45 degree tracks) are placed on those layers which have been assigned horizontal tracks. Obviously, no vias are placed during this pass.

Vertical

During this pass, vertical tracks (with some short 45 degree tracks) are placed on those layers which have been assigned vertical tracks. Once again, no vias are placed during this pass.

L-Route (1 via)

An L-route occurs when one track is laid on one layer and another on a different layer and the tracks are joined by one via.

Z-Route (2 vias)

A Z-route consists of three tracks and two vias which form a Z shape.

C-Route (2 vias)

A C-route consists of three tracks and two vias which form a shape like the letter C.

Any node (2 via)

In this pass either a Z or C route can be assigned and two vias are placed.

Any node (3 via)

In this pass three vias can be placed.

Via minimisation

This pass does not make any more connections, it simply tries to reduce the number of vias on the board. Tracks may be swapped from the top layer to the bottom layer, or vice versa, in an attempt to reduce the number of vias.

You should initially route the board with this option disabled (i.e. 'off') since it may change tracks that you have placed manually.

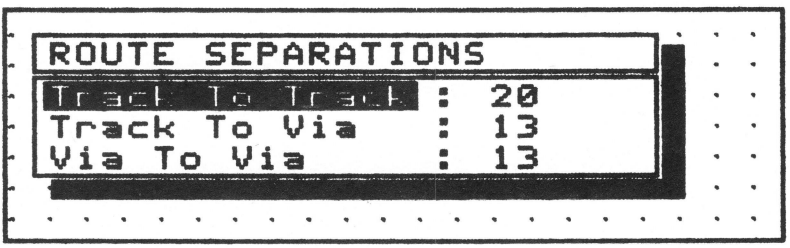
Therefore, as a general rule you should set up the router with the *Process Pre-Routes*, *Any Node (3 via)* and *Via Minimisation* set to 'off'. The *Process Pre-Routes* option should be set to 'on' if you have placed any tracks manually prior to routing.

Separation setup

The *PROTEL-AUTOTRAX* router gives you a great deal of flexibility in the design rules you build into the route.

You can change the clearances between tracks and vias as well as the width of tracks.

If you choose *Netlist Route Separation Setup* you will be able to change the track to track clearance, the track to via clearance and the via to via clearance.



Each of these has a default of 20 mils. You may, however, have a particularly dense board and want to be able to route two tracks between pads rather than one.

Assume that you have 50 mil pads placed 100 mils apart. This gives a clearance of 50 mils between pad edges. A 12 mil track with a 13 mil separation means that only one track will fit between pads and still obey the design rules.

Change each of the separations to 10 mils and change the track width (in *Variable Setup*) to 10 mils. The tracks are now narrow enough for two to fit between the pads with the required 10 mil separation!

Setting the router variables

The *Netlist Route Variable Setup* option allows you to change three important variables which affect the routing of a board; the route grid size, the track size and the via size.

The default route grid size is set to 25 mils. This provides a track-to-track clearance of 13 mils if 12 mil tracks are being placed. If you require a greater separation choose a larger figure.

The default track size is set to 12 mils. If you need two tracks to be laid between 50 mil pads, change the track size to 10 mils and the track separation to 10 mils.

You can change the size of vias which are placed during routing. The default size is 50 mils.

Routing the board

The *Netlist Route Board* is the most powerful of the routing options. With it you can route a partially completed board or an entire board using a previously created netlist.

The process is best explained via an example, so at this stage you should go through the exercise of routing the demonstration PCB supplied on the Library disk.

Before you begin a route make sure you have completed four tasks.

1. Check each of the *Netlist Route* setup options to make sure that the track size and the clearances are correct. In this case you can accept each of the default values.

2. Load in the PCB which is to be routed. In this case load the file PLACED.PCB.

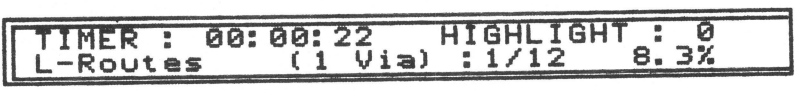
3. Load in the netlist file which contains the information about each of the net connections. In this case load in the file called DEMO.NET.

4. Set up any keep-outs that you want to use to protect areas from transformers, drill holes etc.

You can get visual feedback about the netlist by choosing the *Netlist Show All Nets* option. A set of thin white lines will be displayed showing the connections to be made. This set of connections is called a 'ratsnest' and is, essentially, the information which will be used to automatically place the tracks.

You are now ready to route the board. Choose *Netlist Route Board.* You are prompted to supply a filename for the router results. Type in the name and press «ENTER» or click «LEFT MOUSE».

The routing now takes place automatically using the information from the netlist. Notice that on the status line and the prompt line information is provided about the type of connection being made.



Reading from the left across the status and prompt lines, the following information is displayed. The first field shows the time taken to route the board so far. The time is shown as HOURS:MINUTES:SECONDS. The next number, labelled 'HIGHLIGHT' is the Recursion Depth Counter. This shows how deeply the program is recursing to work out the connection. The first field on the prompt line shows the routing pass being executed. You can expect to see messages such as 'MEMORY ROUTE' or 'ANY NODE (2 VIAS)' in this field. On the right hand side

is the ratio of the number of nets paced to the total number of nets and then the same figure given as a percentage. The higher this figure the closer the router is to placing all tracks.

The router takes each net from the netlist and attempts to make a connection for each of the passes which have been activated. The normal order for the passes is as follows:

Memory routes

Horizontal

Vertical

L-routes

Z-routes

C-routes

Any node (2 vias)

3 vias

Any node (3 vias)

Each of these is explained in the section above entitled 'Setting up the Router Passes'.

As the routing progresses you will see thin horizontal and vertical lines flashing on the screen showing you where the router is attempting to place tracks. It is interesting to watch the strategy adopted by the router during the Z-route and the C-route stages.

If you want to interrupt the router at any stage, press any key (except the «ENTER» key) or click «RIGHT MOUSE». A window will open asking you to 'CONFIRM ABORT ROUTER'. Choose *Yes* to finish routing and *No* to continue. An automatic save is carried out at the end of the routing pass.

When the route has been completed or the router cannot place any more tracks the message 'AUTO-ROUTER FINISHED' will appear in a window at the top of the screen. Note the percentage complete value at the right hand side of the status line so that you know how many nets you need to place manually.

Press «ENTER» or click «LEFT MOUSE» to continue. The route menu reappears.

Now you are faced with the problem of what to do with the remaining nets which must be routed. There are a number of approaches you can adopt.

The first thing that you should do is check the board for obvious areas of congestion. You may have placed a component in an unsuitable place. The first step in overcoming this problem is to return to the unrouted PCB and move the key component in that area to a different part of the board. Don't be afraid to experiment - it does not take very long to re-route a board in the majority of cases!

If that does not produce a satisfactory result move other components which may free up the congested area and try again. Eventually you will reach a stage where the only remaining action is manual routing.

You should manually re-route before you attempt the *Via Minimisation* pass because this pass may change tracks from the top to bottom layer and vice versa. This action may tend to block out some of the possible manual routes. Also it can reduce the number of nodes which you can connect to.

Remember when you are manually routing the remaining connections that you should maintain conventions such as laying tracks horizontally on the top layer and vertically on the bottom layer.

Routing a single connection

Choose *Netlist Route Connection* to route a particular connection which you have highlighted using *Netlist Show Nets*. The message 'SELECT CONNECTION' is displayed on the prompt line. You should then move the cursor to the connection you want to route, then press «ENTER» or click «LEFT MOUSE».

The router then takes over and performs each of the passes that you have enabled.

If it is possible, the connection will be routed. The prompt 'SELECT CONNECTION' now indicates that you can route another connection. Press «ENTER» or click «LEFT MOUSE» to return to the router menu.

Routing a connection manually

If you want to route a connection manually choose *Netlist Route Manual*. You often need to choose this option after an automatic route which has not achieved 100% completion.

You are prompted to 'SELECT CONNECTION'. Move the cursor to the highlighted connection and press «ENTER» or click «LEFT MOUSE».

The message 'PLACE TRACK' now appears on the prompt line. A track will be laid from the closest node to the cursor position when you press «ENTER» or click «LEFT MOUSE». You will notice that the cursor is 'tied' to the connection highlight to ensure that you connect the tracks to the correct pads.

A via will be placed every time you change layers with the «*» key if the *Via Mode Auto* option is set to 'on'.

The 'PLACE TRACK' prompt continues to appear until the connection has been fully routed.

Routing a net

Choose *Netlist Route Net* to route a named net rather than a connection. You will be asked to enter the name of the net (or you can choose it from a list if you enter a «?»). Once the net is chosen, the process is identical to that for *Netlist Route Connection.*

Using the pad to pad router

To route a connection between two pads automatically, choose *Netlist Route Pad to Pad.* The message:

ROUTE: SELECT FIRST PAD

appears. Point to the first pad you need to connect and press «ENTER» or click «LEFT MOUSE».

The message now changes to:

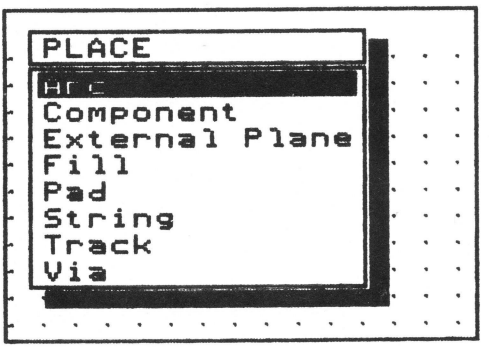
ROUTE: SELECT SECOND PAD

Move to the second pad and press «ENTER» or click «LEFT MOUSE».

The router now takes over and determines a path for the tracks. Sometimes it will also place a via to route that track to the other side of the board. When it has finished placing tracks you will be prompted to select the first pad for the next route. Choose another pad if you want to route another connection. If you have finished routing, press «ESC» or click «RIGHT MOUSE» to exit from the ***Pad to Pad*** routine.

21 PLACING ITEMS ON A BOARD

The *Place* option allows you to manually place items such as arcs, components and tracks on a board. It also allows you to place special shielding tracks called an external plane. As a general rule, you should ensure that you are on the correct layer before you place items using this option.



Placing an arc

You may need to place an arc if you are creating your own component or if you want to produce an arc-shaped outline on the board.

Choose *Place Arc*. A prompt appears on the status line asking you to place the arc. Move the cursor to the position you want the arc to appear at and click «LEFT MOUSE» or press the «ENTER» key.

A white circle appears and you are asked to 'SELECT ARC RADIUS'. To do this, use the «RIGHT» cursor key to increase the radius and the «LEFT» cursor key to decrease the radius. If you need to change the radius significantly, then hold the «SHIFT» key down and use the «RIGHT» and/or «LEFT» cursor keys. The change in radius will be ten times greater.

When you have the radius you require, press «ENTER» or click «LEFT MOUSE», and you are then prompted to select the arc segments. The circle is divided into four segments. One is from 12 o'clock round to 3 o'clock and is under the control of the «UP» cursor key. The next is from 3 o'clock to 6 o'clock and is under the control of the «RIGHT» cursor key. The 6 to 9 segment is controlled by the «DOWN» key and the remaining segment is controlled by the «LEFT» key. Pressing any of these keys results in that segment of the arc being deleted. Pressing the same key again restores the deleted segment.

As an example, if you want an arc which extends from 3 o'clock around to 9 o'clock in the clockwise direction, press the «LEFT» key and the «UP» key.

When you press «ENTER» or click «LEFT MOUSE» you are given an opportunity to move the arc if it is not positioned correctly.

Placing a component

When you choose ***Place Component*** you are prompted to indicate where you want to place the component. Move the cursor to the place required and click «LEFT MOUSE» or press «ENTER». An input window opens asking you to enter the name of the component. If you have not yet placed a component the window contains a question mark.

You can now type in the component name or else press «ENTER» or click «LEFT MOUSE» to see the list of components which are available.

Move the selection bar to the required component. If the component does not appear in the window press «PGDN» to see more. Press «ENTER» or click «LEFT MOUSE» to select the component.

The prompt:

COMPONENT DESIGNATOR

appears.

You must now supply a component designator. A component designator usually consists of a letter such as C or J or R followed by a number. Thus you may decide to call an edge connector J3 or a capacitor C4. If you do not supply a designator *PROTEL-AUTOTRAX* will assign one

starting at A1 then A2 and so on. If you specify a designator which already exists the message:

COMPONENT DESIGNATOR IS ALREADY USED

appears.

After you supply the designator you are asked to input a comment. Remember that any comments will appear on the layer selected in *Setup Component Text*. It is a good idea to keep them as short as possible.

Once you have entered the comment the prompt:

MOVING COMPONENT

appears. The component can now be moved to another position if required. For details of how to rotate the component refer to page 66. When it is in the correct position press «ENTER» or click «LEFT MOUSE».

Note that when you have placed the component, you are prompted to place the next one. If you do not want to place another component press «ESC» or click «RIGHT MOUSE» to exit from the option.

Creating an external plane

If you need to provide extra shielding on a particular layer (perhaps to conform to local RF emission standards) then it is possible to fill an area with cross-hatched wide tracks - even if that area has already been routed.

Choose ***Place External Plane***. The message:

SELECT START POINT ON POLYGON

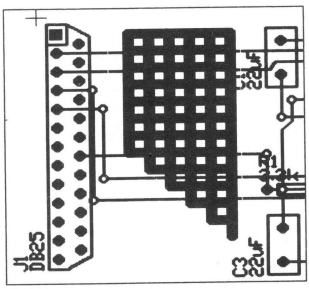
appears on the prompt line. You can now define any area in the shape of a regular or irregular or convex or concave polygon to be filled. Press «ENTER» or click «LEFT MOUSE» to start defining the polygon.

Now the prompt changes to:

SELECT POINT ON POLYGON

and you can now indicate each of the vertices of the polygon. When you have finished marking the polygon press «ESC» or click «RIGHT MOUSE».

The status line now changes to indicate the time taken to fill the area and the recursion depth counter will appear on the right hand side. You will see the area you have defined being filled with the cross-hatched pattern.



The tracks are placed on the currently selected layer so make sure that you choose the correct layer before invoking this option.

If you do not fully complete the polygon, then the last vertex you mark is joined to the start point to complete it.

Placing a fill

If you need to place a rectangular area of copper then choose the ***Place Fill*** option.

The message:

SELECT FIRST CORNER

appears on the prompt line at the bottom of the screen. After you select the first corner an outline of the rectangle appears to help you place it accurately. Click «LEFT MOUSE» or press «ENTER». Now the message:

SELECT SECOND CORNER

is displayed. Move the cursor to the second corner and click «LEFT

MOUSE» or press «ENTER». As soon as you select the second corner the rectangular area is filled.

If you select the second corner in the same place as the first corner you will hear a beep and you will be asked to respecify both the first and second corners again.

Placing a pad

To place a pad, select the ***Place Pad*** option and move the cursor to the required position on the board then click «LEFT MOUSE» or press «ENTER».

The shape of the pad which is placed on the board is determined by the setting in the ***Current*** menu. The default type is called Round62. If you require a pad of a different shape (say, a square pad), then change it in the ***Current*** menu and then go back to the ***Place*** menu. You can also choose Moire or Cross-Hair targets to help you line up plots.

Remember that pads belong to every layer, therefore you do not have to worry about which layer you are on before placing a pad.

Placing a string

To place a string choose the ***Place String*** option. A window opens which allows you to enter the string to be placed. Enter the string then press «ENTER» or click «LEFT MOUSE».

The string appears on the board in inverse and you can now move it to the place required. Use the cursor keys or the mouse. For details of how to rotate the string refer to page 66.

The size of the string depends on the settings in the ***Current*** menu.

After the string is placed you get the opportunity to specify another. When you have finished placing strings, click «RIGHT MOUSE» or press «ESC».

Placing tracks

Individual connections can be laid by using the ***Pad to Pad*** routing option in the ***Netlist Route*** menu. If, however, you need to lay tracks manually for some reason then choose ***Place Track.***

The message:

SELECT TRACK START POINT

is displayed on the prompt line. Move to the point where you want the track to begin and press «ENTER» or click «LEFT MOUSE». Now when you move the mouse or use the cursor keys a thin line is drawn outlining where the track will be laid. When the cursor is at the point that you want to lay the track press «ENTER» or click «LEFT MOUSE». The track is drawn at the width specified in the *Current* menu.

If you now move the cursor to a different position the track will be continued, and so on. If you click «RIGHT MOUSE» or press «ESC» then you can move to a different spot on the board and lay another track.

To stop laying tracks press «ESC» or click «RIGHT MOUSE» when the prompt 'SELECT TRACK START POINT' appears.

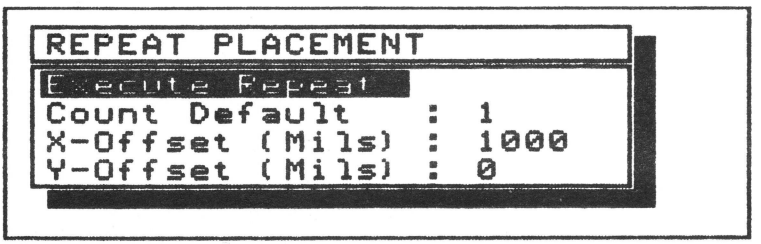
Tracks can only be drawn as vertical, horizontal or 45 degree diagonal lines. This has been set as it is regarded as good practice in the industry to only draw tracks this way. If, however, you need to draw a track at some other angle, select the *Current* menu and set the *Toggle Ortho Mode* option to 'off' before you lay the track.

Placing a via

The process for placing a via is identical to that for placing a pad. Simply choose *Place Via* and move to the place where you want to place the via then press «ENTER» or click «LEFT MOUSE». The size of the via is determined by the setting in the *Current* menu.

22 REPEATING ACTIONS

The *Repeat* option lets you automatically place a component, track, pad or via a number of times at predetermined places. This is useful if you have a number of items which you want to place in a regular pattern.



When you choose *Repeat*, the window shown above opens allowing you to change the count default, the X-Offset and the Y-Offset. If you choose *Count Default* you can change the number of times the previous placement is repeated. The X-Offset and Y-Offset determine the direction in which the placements will be made.

When the option is executed, the previous placement is duplicated in a regular pattern from the position of the previous placement. Thus if you place a pad at X:1000 Y:800 and set a *Count Default* of 3 and an *X-Offset* of 200 and a *Y-Offset* of 50, the three pads will be placed at X:1200 Y:850, X:1400 Y:900 and X:1600 Y:950 when you execute it. The cursor will remain at X:1000 Y:800.

The offsets can also be negative if you want to duplicate down the board (make the Y-offset negative) or towards the left edge of the board (make the X-offset negative).

To implement a repeat operation choose ***Repeat Execute Repeat***. You will be prompted to enter the Count Default again just to make sure that you really want to repeat the operation the number of times you have specified. You are then prompted to:

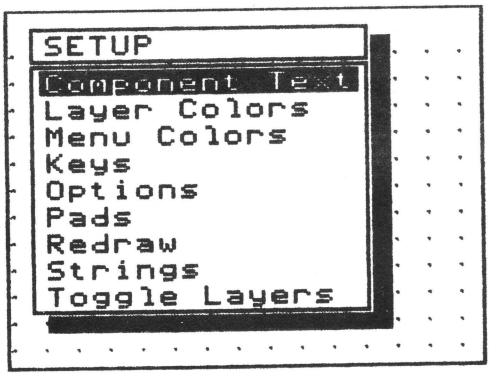
CONFIRM MULTIPLE REPEAT PLACE

Choose *Yes* to execute the repeat and *No* to return to the ***Repeat*** menu.

If you use the ***Repeat*** option to copy a track, you should be aware that only the last track placed will be copied. The option does not allow you to copy an entire connection. If you want to copy a set of tracks, use the ***Highlight Duplicate*** option.

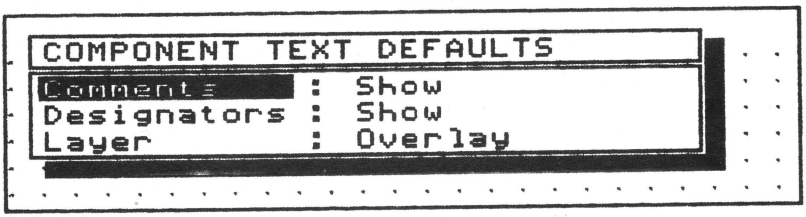
23 CHANGING THE SETUP OPTIONS

The *Setup* menu allows you to change many of the 'default' parameters which exist each time you start *PROTEL-AUTOTRAX*. If you change them, they will be saved as the default parameters.



Controlling component text

Setup Component Text lets you choose the layer on which the component text will be placed. It also lets you hide the component text if required. When you choose the option the following menu appears.



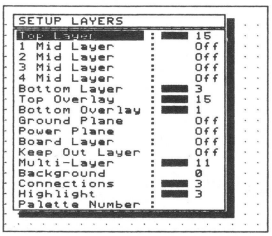
The *Comments* option lets you choose to *Hide* or *Show* the designator comment, while the *Designator* option lets you hide or show the designator of the component.

When you choose *Setup Component Text Layers*, a window opens giving you the choice of putting the text on the copper layer or the overlay. If you are placing SMD devices on the bottom layer and you choose *Copper* then the text is placed on the bottom layer. If you choose *Overlay*, then the text is placed on the bottom overlay.

If you are placing multi-layer components then the text will either be placed on the top layer or the top overlay depending on whether you choose *Copper* or *Overlay* respectively.

Choosing layer colors

Choose *Setup Layer Colors* to change the layer colors if the colors on your screen are not suitable. There is some variation in the quality and adjustment of screens, so you may need to adjust the colors to provide adequate contrast between the background layer and the various other layers you might be using.



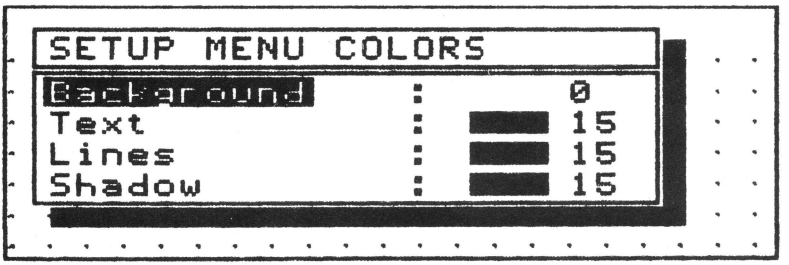
The number of colors available for each layer varies depending on the type of monitor you are using. If you have an EGA screen there are sixteen colors numbered 0 to 15. If you have a CGA screen there are only be four colors numbered 0 to 3. If a layer has been turned 'off' in *Setup Toggle Layers*, the word 'Off' appears next to the layer in the menu. You cannot assign a color to a layer which is turned off. The layers are explained in more detail in the section called 'Turning layers on and off'.

If you are using a CGA monitor you can change the colors used for the display by choosing *Palette Number* from this menu. The option is inoperative if you are using an EGA monitor.

Move the selection bar to the layer you want to change then click «LEFT MOUSE» or press «ENTER» to change the color. Exit from this menu by pressing «ESC» or clicking «RIGHT MOUSE».

Selecting menu colors

You can also change colors in the pull-down menus. The background color, the color of the text and lines and the color of the shadow around the menu can all be altered.



Choose *Setup Menu Colors*. The colors can then be changed in the same way as for the layer colors. Move the selection bar to the one you want to change then click «LEFT MOUSE» or press «ENTER» until the required color is displayed.

Be careful to ensure that you do not choose a color for the menu which is the same as the background layer color. *PROTEL-AUTOTRAX* does not do any cross-checking with the layer colors to stop this happening.

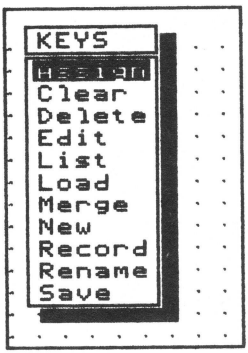
Assigning functions to keys

PROTEL-AUTOTRAX allows you to assign sequences of commonly used keystrokes to function keys to speed up operations. If you regularly repeat a sequence of keystrokes, such as when you are placing pads, then they can be defined as a **macro** and assigned to a particular function key.

Once assigned, the macro can be invoked simply by pressing one key or a simple key combination such as «SHIFT» or «CTRL» or «ALT» followed by a function key.

In the menus a function key labelled F1 is designated by <F1>. Pressing the «SHIFT», «ALT» and «CTRL» keys while holding down the <F1> key is designated by <SHIFT-F1>, <ALT-F1> and <^F1> respectively.

PROTEL-AUTOTRAX is supplied with a default set of macros which you may want to use sometimes.



There are a number of menu options available to help you define your own macros. Each of these options is now be explained briefly. Remember that Chapter 6 contains a tutorial explaining how to define macros.

Assign

Choose *Setup Keys Assign* to reassign a macro from one function key to another. You may need to do this if you find you are using a macro often, yet have assigned it to a two-key combination such as «CTRL» «F5».

You are asked to select the macro by name or by key. When you choose one of these options a window opens listing either the names of all macros currently assigned to keys or the keys which currently have macros assigned to them. Select the macro to be reassigned. An input window

opens asking you to select the function key to which the macro is to be assigned.

Press the function key or function key combination that you want to assign the macro to.

You are returned to the selection window. If you have finished assigning macros press «ESC» or click «RIGHT MOUSE» to return to the *Keys* menu.

Clear

Choose the *Setup Keys Clear* option clear all macros from the memory. Although macros are cleared from the memory the disk file is not affected.

You are asked to confirm that you want to clear all key macros. Choose *Yes* to clear the macros and *No* to leave them intact.

Delete

Choose *Delete* to delete one or more macros from the memory. The macros are not deleted from the .KEY file unless you choose to save the new set of macros.

You are asked to select macros by name or by key. Once you have selected a macro that you want to delete you are asked to confirm the deletion. Choose *Yes* to proceed and *No* to stop the macro being deleted.

Edit

Once you have defined a macro you can add to it or delete parts of it. Select the macro that you want to change and an edit window opens. The name of the macro and the function key it is assigned to is displayed at the top of this window.

The window displays the representations of each of the keystrokes that comprise the macro. If you press the «BACKSPACE» key you will delete items in the macro. If you press any other key you will add that keystroke to the macro.

You can press other function keys which themselves contain macros at this stage.

Press «ENTER» or click «LEFT MOUSE» to finish editing the macro.

List

You can list the current macros by name or by key. The macro names or key names are displayed in an information window.

Load

Choose this option by moving the selection bar to highlight the word 'Load' then pressing «ENTER» or clicking «LEFT MOUSE». The reason is that the *List* option also begins with L and is before Load in the list.

An input window opens with the default value of '*.KEY'. To load a new key file either type in the name of the file, or press «ENTER» or click «LEFT MOUSE» to see a list of key files.

Select a file then a window opens asking you to confirm that you want to load it. Choose *Yes* to load and *No* to return to the *Key* menu without loading the file.

New

There are two methods you can use to define a new key macro. The first is by choosing *Setup Keys New*. An input window opens and you can enter the name you want to give the new key macro. Another window then opens asking you to choose the key to which the macro will be assigned. A default value (which is the next available key) appears. If the default value is not acceptable, press another function key combination.

A macro edit window then opens allowing you to type in the keystrokes that you want to assign. Press «ENTER» or click «LEFT MOUSE» to exit from this window. You are then prompted to enter another new key macro name. If

you have finished defining macros press «ESC» or click «RIGHT MOUSE».

Record

The second method of defining a key macro is by recording it. This option allows you to work through the macro while seeing the effect each keystroke has on the board. Mouse movements are not recorded.

Choose *Setup Keys Record.* You are prompted to supply a name and a key as in the New option. You will then hear a beep and you are returned to the board where you can then simulate the set of keystrokes that you want to store in the macro. When you have completed the keystrokes press «CTRL» «END» to turn the 'recorder' off. You are then be returned to the *Key* menu.

Rename

Choose *Setup Keys Rename* to change the name of a macro, rather than the key it is assigned to. You are prompted to supply the name which you want to change and then the name you want to change it to. If the name you try to change it to already exists the message 'KEY MACRO NAME ALREADY USED' is displayed.

Save

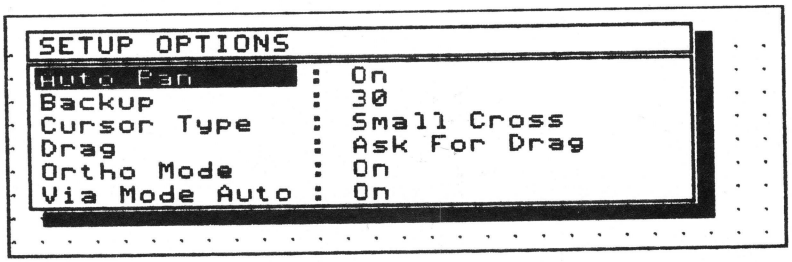
Once you have changed some macros or created one or more new macros you will want to save them in a file so that you can use them again. You can either save them under the default name of TRAXEDIT.KEY or choose a name of your own.

If you try to save them under a name that already exists the warning:

KEY MACRO FILE EXISTS. OVERWRITE?

is displayed. Choose *Yes* to delete the existing macro file. Otherwise choose *No*. You can then enter another name.

Setting options



The *Setup Options* menu allows a number of important features of the program to be changed.

The *Auto Pan* option can either be 'on' or 'off'. If it is 'on' then moving the cursor to the edge of the screen moves you to a new section of the board. This is a nice feature if you have a powerful microcomputer which can redraw the screen quickly.

Sometimes, however, you may want to do a lot of work on one section of the board. Turning *Auto Pan* 'off' means that bumping the mouse or inadvertently moving the mouse to the edge of the screen will not cause it to be redrawn.

The *Backup* feature is vital if you want to avoid losing important work. It is set to do an automatic backup every 30 minutes. If you are working with a small board you may wish to decrease this to, say, 10 minutes.

Choose *Setup Options Backup.* An input window opens. The default name of:

TRAXEDIT.ABK

is displayed. If you want to change the name which is assigned to the backup file then enter it in this window. The current time between backups is also displayed. Type in the new value then click «LEFT MOUSE» or press «ENTER» to close the window.

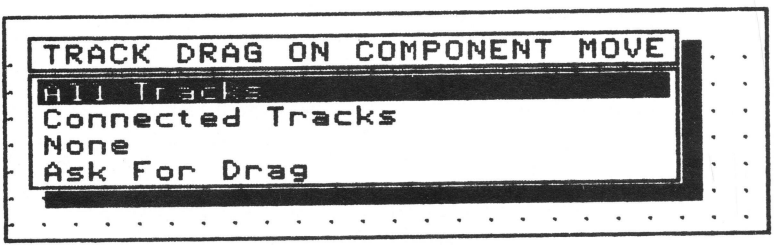
If you lose your work you can rename the backup file with the DOS RENAME command. If the backup file is called TRAXEDIT.ABK and

is stored in the \PCB directory and you need to rename it to be, say, BOARD.PCB type the following command at the DOS prompt.

```
RENAME \pcb\traxedit.abk \pcb\board.pcb
```

The *Cursor Type* option lets you change the cursor from a small cross to a large cross or to an arrow. Many users prefer the arrow because it is easier to locate on a board with many components and tracks. When you choose this option a menu appears with the three choices. Move the selection bar to the one you require and click «LEFT MOUSE» or press «ENTER».

The *Drag* option lets you choose what happens to tracks when a component is moved from its original position on the board. A menu provides four options.



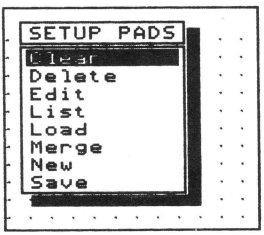
You can decide that all tracks should be affected, only the connected tracks, no tracks at all or else the final option causes a question to be asked each time so that you can make a decision based on the type of component and the number of tracks that are likely to be involved.

PROTEL-AUTOTRAX normally allows you to produce tracks in orthogonal mode. This means that they can only be laid horizontally, vertically or at 45 degrees. Such an approach is in keeping with good practice in the industry.

Nevertheless, if you decide to do so, the *Ortho Mode* option allows you to turn this option 'off' and hence provide you with the ability to lay tracks at any angle.

The *Via Mode Auto* option provides for automatic via placement when placing tracks. The options are 'on' and 'via'. If the option is 'on' then when you lay a track and switch to another layer then lay another track, a via will automatically be placed. If the option is set to 'via' then you must place your own vias.

Defining pad types



This option contains some powerful features which makes it easy for you to define your own pad shapes, and to manage the storage and retrieval of pad shapes. This is achieved through the use of pad definitions.

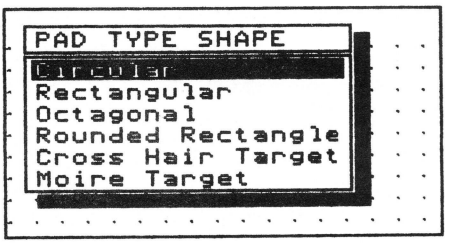
The *Clear* option clears all currently defined pad types from the memory. Only use this option if you really do want to erase all definitions.

If you just want to remove some of the types, then choose the *Delete* option. A window opens showing all the pad types which are currently available. Select the type you wish to delete. You are asked to confirm your choice. If you choose *Yes* the definition is deleted. If you choose *No* the window reappears and another choice can be made.

Click «RIGHT MOUSE» or press «ESC» to return to the *Setup Pads* menu.

The *Edit* option allows you to change an existing pad definition. When you choose the option a window opens listing all the active pad definitions. Select the one you want to edit and the attributes of the pad are displayed. Each of these can be changed.

If you want to change the shape, for example, move the selection bar to highlight the shape and either click «LEFT MOUSE» or press «ENTER». A window opens with each of the possible pad shapes listed.



Choose the one you want and then you are returned to the *Edit* menu. Each of the other attributes can be changed in a similar way.

There is a lot of checking done behind the scenes to ensure that you do not create a pad with silly attributes. For example if you specify a Surface Mount Device (SMD) then it would not be sensible to specify a hole size. Similarly it would not be sensible to specify a hole size that is greater than the diameter of the pad. If you do try to specify a hole size which is greater than the pad size, the program automatically clips the hole size to be one less than the pad size.

The *List* option gives a list of all the pad types which are currently held in the memory.

The *Load* option allows you to load pad definitions from a file. The filename must be entered in the input window or else type *.PAD and a window opens showing all files which hold pad definitions. If this option is chosen then the pad types which are currently in the memory will be overwritten.

If you do not want to overwrite the existing definitions, but simply want to add a few more types to those already in the memory then choose the *Merge* option. This option works in the same way as the *Load* option except that the definitions are added to those currently in the memory rather than overwriting them.

One problem that you may have with the *Merge* option is that *PROTEL-AUTOTRAX* can only store a maximum of 48 pad definitions in the

memory at one time. Thus if you are going to merge a file you must be confident that the total of the definitions in the memory and those in the file does not exceed 48.

The *New* option allows you to define a completely new pad definition. When you choose the option you are asked to supply a name for the new definition. Once again, you must first ensure that there are not already 48 in the memory.

Enter the name and click «LEFT MOUSE» or press «ENTER». A window opens showing the possible attributes of the pad. These will have been supplied from the definition which is shown as the default in the *Current* menu.

The process of defining the type of pad is exactly the same as that for editing a pad. Simply move the selection bar to highlight the attributes that require changing and select the new ones.

The final option allows you to *Save* your pad definitions. Choose *Setup Pads Save*. Enter a name for the file and either click «LEFT MOUSE» or press «ENTER». The file is saved with a .PAD extension.

Choosing the redraw quality

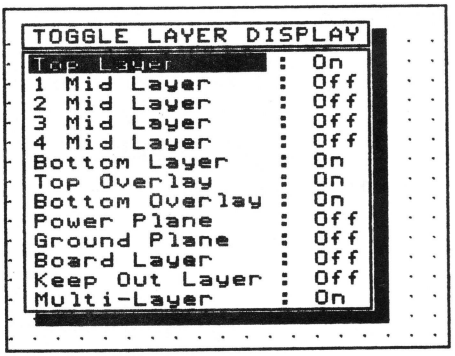
This option allows you to choose the quality of each redraw for arcs, fills, pads, strings, tracks and vias. Each one can be set to either draft or final.

Setting string defaults

The term string refers to the characters that are used as comments and component designators as well as the free text that can be placed on a board.

The default value for each is 60, but it is possible to change each by typing in a value up to 1000. A maximum value which is greater than 560 mils is not recommended.

Turning layers on and off


TOGGLE LAYER DISPLAY
Top Layer : On
1 Mid Layer : Off
2 Mid Layer : Off
3 Mid Layer : Off
4 Mid Layer : Off
Bottom Layer : On
Top Overlay : On
Bottom Overlay : On
Power Plane : Off
Ground Plane : Off
Board Layer : Off
Keep Out Layer : Off
Multi-Layer : On

This important option allows you to decide whether a layer will be displayed or not. As there are 13 choices available it is not a good idea to have them all turned on at the same time. Simply select the layer and change it from 'on' to 'off' or 'off' to 'on'.

A brief description of each of the layers is provided below.

Top layer

This layer is sometimes known as the component side and is the layer on which you place components and tracks which cannot be routed on the bottom layer. Surface Mount Devices can also be placed on this layer.

Mid Layer 1,2,3,4

These are signal layers used in multi-layer boards. You can route tracks on these layers.

Bottom Layer

This layer is sometimes known as the solder side. It is used for routing tracks. Surface Mount Devices can also be placed on this layer.

Top overlay

This layer is often used for placing text.

Bottom overlay

This layer is used for text which may be necessary to explain features on the solder side.

Ground Plane

The ground plane is the plane which is used to shield sensitive sections of the circuit and to minimize radiated energy. It is a separate plane of copper sandwiched between the other layers. There are holes in the copper wherever leads pass through to other layers so that they do not contact with the copper.

Power plane

The power plane is very similar to the ground plane. It is used when large currents are needed.

Board layer

The board layer provides a similar function to the keep out layer. It is used, for example, when you want to show the edges of the board in a plot, but do not want to plot all the other 'keep outs'.

Keep out layer

This is a very important layer which is used to define keep out areas on the board. It is often used to define the limits of the board. To do this, turn the keep out layer on and select that layer by pressing the «+» key on the keypad.

Now lay tracks right around the limit of the board. Since the keep out layer affects every layer, the router will not be able to place tracks outside this border.

You can also define keep out areas in places where you want to drill through the board.

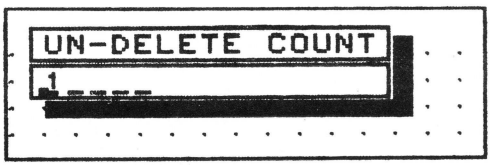
Multi-layer

With this layer you can define the color for multi-layer items such as pads and vias.

24 USING THE UN-DELETE OPTION

The *Un-delete* option is your safeguard against accidental deletion of carefully placed items.

When you choose *Un-delete*, a window opens asking you to specify the 'UN-DELETE COUNT'. This is the number of items you have deleted that you want to restore to the board.



PROTEL-AUTOTRAX starts restoring deleted items, *starting from the most recently deleted item.*

A simple example

You may gain a clearer understanding of how this option operates by working through the following example.

Move to a blank area on the board and choose *Place Component*. Place, in order, a DIP4, DIP6, DIP8, DIP16 and DIP40. Arrange them so that they are side by side on the board. Don't worry about the designators or comments.

Now exit from the *Place* option and choose *Delete Component*. Delete each of the components you have just placed starting with the DIP40 pattern and working back to the DIP4 pattern.

Now choose *Un-delete*. Input the value 2 for the 'UN-DELETE COUNT' and press «ENTER» or click «LEFT MOUSE».

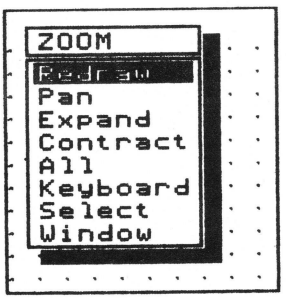
You will see that the DIP4 and DIP8 patterns have been restored because these were the two that were deleted most recently.

Now choose *Un-delete* again. This time specify an 'UN-DELETE COUNT' of 1. The DIP8 pattern will now be restored.

The 'stack' of deleted items is capable of storing up to 5000 items.

25 EXPANDING AND CONTRACTING THE DISPLAY

The *Zoom* option in the MAIN MENU can be used to expand and contract the display. Sometimes you will need to have a view of the entire board (e.g. when you are highlighting nets) and other times you will need to magnify a section to make minor placement adjustments (e.g. accurate placing of pads).



Redrawing the screen

Choose *Zoom Redraw* to redraw the current screen at the current magnification. This is useful if you have just deleted, say, some tracks or components.

An alternative is to press the «END» key at any stage.

Moving around the board

If you have the *Auto Pan* option set to 'on', then every time the cursor reaches the edge of the screen it 'pans' to the next screen.

Choosing *Zoom Pan* makes the current cursor position the centre of the screen. This is particularly useful if you have turned the auto pan option 'off' by choosing *Setup Options Auto Pan Off*.

Expanding the display

If you need to work at a higher magnification then choose *Zoom Expand*. The display will be momentarily blanked and then redrawn showing more detail of the features on the board. The expanded display is centered around the current cursor position unless it is on the very edge of the board.

Repeat the process and you will notice that the display is expanded further. There are six zoom levels available. At the lowest level one screen pixel represents 100 mils whereas at the highest level one pixel represents 1 mils.

Note that pressing the «PGUP» key zooms the display up one level. Pressing «PGUP» a number of times quickly causes it to zoom straight to the higher level without drawing the intermediate levels.

When the highest level of magnification has been reached, pressing «PGUP» or choosing *Expand* no longer has any effect on the screen.

Contracting the display

This option provides the opposite facility to the *Expand* option. Each time *Contract* is chosen the screen is redrawn but contracted one level.

Note that pressing the «PGDN» key also contracts the display one level, and pressing it a number of times quickly has the same effect as with the «PGUP» key.

Fitting the board on the screen

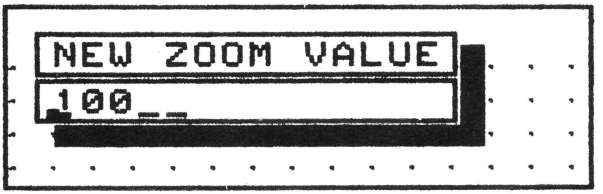
Choose *Zoom All* option to redraw the display so that the entire board can be seen on the screen.

If the board size has been changed since you started the editing session then you should first choose *Information Board* so that *PROTEL-AUTOTRAX* can work out the scaling necessary for it to fit exactly onto the screen.

Specifying a zoom factor

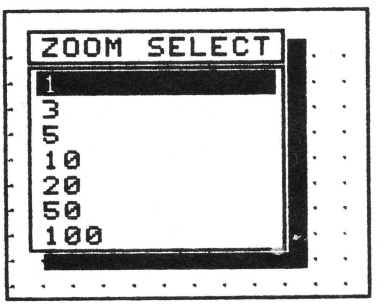
If you want to directly specify a zoom factor, choose the *Zoom Keyboard* option. An input window opens, usually with a default value

in it. Type in the value you require then press «ENTER» or click «LEFT MOUSE». Remember that the larger the number you enter, the smaller the board will appear on the screen.



Selecting a zoom value

This option allows you to select from one of the seven pre-set zoom values.



Move the selection bar to the magnification you require and click «LEFT MOUSE» or press «ENTER». The screen is redrawn at the zoom level you selected.

Specifying a zoom window

If you want to inspect a particular section of the board in more detail choose the *Zoom Window* option.

You are prompted on the status line to select the first corner of the window. Click the «LEFT MOUSE» button or press «ENTER» when the cursor is at the corner of the area you want to select. Move the cursor to the opposite corner of the rectangular area you want to inspect. A box

appears outlining the window. The prompt now reads 'SELECT SECOND CORNER'. Click the «LEFT MOUSE» button or press «ENTER» to select the second corner.

The window now zooms to fill the screen.

UTILITIES SECTION

26 USING THE *TRAXPLOT* PROGRAM

The *TRAXPLOT* program has been designed to allow you to produce quality artwork using a conventional pen plotter or a photoplotter and to produce draft quality check plots. It also allows you to produce files which will drive Gerber photoplotters and Excellon NC Drills.

One of the features of this program is that the external protection device is not required. This allows you to use the program on a low-cost PC connected to a printer or plotter, thus freeing up more expensive equipment for the task of designing printed circuit boards.

Starting the *TRAXPLOT* program

If you are using a dual floppy disk system, put the *Plot* disk into drive A: and the disk with the .PCB file that you want to print into drive B:.

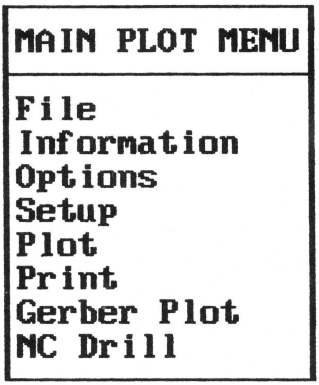
If you are using a hard disk drive you should make sure that you are in a directory from which the *TRAXPLOT* program is accessible.

Type: **TRAXPLOT** then press «ENTER».

When the program is activated, a menu called the MAIN PLOT MENU appears in the top left hand corner of the screen and a **status window** appears towards the bottom of the screen.

The menu is similar to the MAIN MENU in the *PROTEL-AUTOTRAX* program. The **status window** displays the current status of some parameters relating to the plot.

The MAIN PLOT MENU displays a number of options which you need to set up a plot. Each of these are now discussed in detail.



The *File* menu

When you select *File* the following menu is displayed.

There are five options which allow you to manipulate files in a number of ways.

The *Dos* option

This option is identical to the DOS option in the *PROTEL-AUTOTRAX* program. When you choose *Dos* you are asked to confirm the fact that you want to exit to DOS. If you choose *Yes* you can then carry out all the normal DOS functions.

The same restrictions as those mentioned in the *PROTEL-AUTOTRAX* program apply.

When you are ready to continue using the *TRAXPLOT* program, simply type:

EXIT and press «ENTER»

at the DOS prompt and you will be returned to the program.

Loading a file

The *Load* option allows you to load the .PCB file that you want to print or plot. When you choose the option an input window opens asking you to enter the name of the file you want to load. Enter the name then press «ENTER» or click «LEFT MOUSE».

The default value of *.PCB in the window indicates that if you do not remember the name of the file you can press «ENTER» to call up the information window which lists the names of all the .PCB files in the currently selected directory.

When you choose a file and the load begins, a load window will open showing the items being loaded into memory. This window gives you a good idea of the number of components and other items that will be plotted or printed.

When the file has been loaded, the information in the status window is updated to show the name of the new file.

If there are no files available for printing or plotting, the message:

NO FILES FOUND

is displayed.

Changing the path name

If you choose *Path*, you can change the path to the directory containing the .PCB files you want to print or plot.

An input window opens asking for the name of the new path. Type in the path name then press «ENTER» or click «LEFT MOUSE».

If the path doesn't exist then the error 'PATH NOT FOUND' is displayed.

Once you have specified a path, this will automatically be appended to the front of any file specification that you enter in the *Load* option.

Listing files

When you choose the *Files* option, a window opens asking you to enter a mask for the filenames that you want to list. If you enter:

.

then all the files in the currently selected directory are displayed. If you enter something like:

***.PCB**

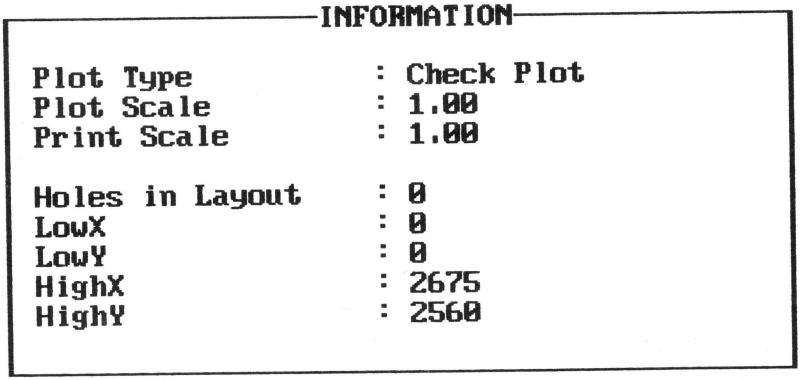
then only the .PCB files are listed.

Quitting

This option allows you to quit the program. Choose *Quit* and a window opens asking you to confirm the fact that you want to quit to DOS. If you choose *No* then you can continue using the program. If you choose *Yes* you are returned to DOS.

Displaying information about the board

If you choose *Information*, a window which displays information relating to the plot or print is displayed. The information in the window is shown below.



Plot Type shows the currently selected plot type such as Ground Plane or Check Plot. The type of plot is chosen through *Options Type of Plot.*

Plot Scale and Print Scale show the scaling factor to be used in the print or plot. The default value for each is 1. These can be changed by choosing *Printer Setup Scale* or *Plotter Setup Scale.*

On the next line the number of holes in the board is displayed. You cannot change this value - it is provided for information only.

The final values define the X and Y coordinates of the bottom left hand corner and the top right hand corner of the board. These values give you an exact measurement of the size of the board to be plotted or printed.

Notice that you cannot alter any of the values in the *Information* screen. Pressing any key or mouse button will return you to the MAIN PLOT MENU.

Choosing the plot options

There are a number of options that you can choose to affect the plot or print. Choose *Option* from the MAIN PLOT MENU and the following menu appears.

SETUP OPTIONS		
Type of Plot	:	Check Plot
Border	:	On
Pads	:	On
Strings	:	On
Title Block	:	On
Vias in Solder Mask	:	On
Pad Hole Size	:	0
Solder Mask Enlargement	:	0
Paste Mask Enlargement	:	0

When you choose the *Type of Plot* option a window opens which lists the eighteen plot options.

The *Check Plot* option allows you to produce a quick multi-layer check plot or check print. If you use a single pen plotter, you may be asked to

change the pen after each layer has been plotted. This could happen if you have assigned different pens to different layers in ***Checkplot Pens***, to ensure that each layer is plotted in a different color.

You can choose to print or plot each of the signal layers and the overlays. You can choose to print or plot the ground plane, but remember that since these planes consist mainly of copper, they will appear in the negative. You can also choose to plot solder masks or paste masks from this menu.

When you choose any of the options from this menu you will notice that the information is updated in the status window.

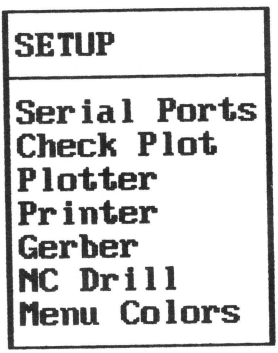
The next four options in the *Options* menu allow you to decide whether the border, pads, strings and title block will be printed or not. Choose the option you want to change and switch it from 'on' to 'off' or vice versa.

The pad hole size for the plot is automatically set to zero. If you want to change this value choose *Options Pad Hole Size*. An input window opens and you can type in the value you require then press «ENTER» or click «LEFT MOUSE».

You can also specify an enlargement factor for the solder mask and the paste mask in the same way.

The *Setup* option

The *Setup* option allows you to choose many of the characteristics of the plot you want to undertake.



Serial ports

When you choose *Setup Serial Ports* a window opens allowing you to choose one of two serial ports. Choose either port 1 or port 2, then click «ENTER» or press «LEFT MOUSE».

You can now change some of the serial port parameters such as the baud rate, the number of data bits, the parity type and the number of stop bits. There is a detailed description of these features in Appendix B of this manual.

Check plot

Choose *Setup Check Plot* to select the layers you want to be output in the check plot.

A window listing each of the layers opens. You can then toggle each layer 'on' or 'off' as required.

Plotter setup

When you choose the *Plotter* option, a window opens allowing you to change a number of parameters relating to the plot. If you choose *Type* a window will open listing the types of plotter for which drivers are available. Select one by highlighting it then pressing «ENTER» or clicking «LEFT MOUSE».

Choose *Device* to select a serial port, parallel port or an output file as the device to which the output will be sent.

The *Scale* option lets you enlarge or reduce a plot and to correct for scaling errors inherent in your plotter.

If you are producing final artwork, you may want the plots to be twice the size of the normal output. To do this, change the *Scale* to 2. If you want to reduce the artwork to half the size, change the Scale factor to 0.5.

The correction values change the scaling in either the X direction or the Y direction. You may need to use these correction factors if your plotter is inaccurate in either the X or Y direction. To check for errors, draw a vertical and horizontal track of exactly the same length. When you have plotted the tracks, measure them carefully. If they are not the same you will need to correct either the X or the Y value.

The X and Y offsets are used to start the plot at a point other than the normal origin. This may be because you want to plot more than one board on the same piece of film, for example. Enter a new value for both the X and Y offsets and the press «ENTER» or click «LEFT MOUSE».

Choose *Orientation* to select 'Normal' or 'Rotated' for the final plot. Choose *Quality* to select 'Draft' or 'Final Artwork' for the plot.

If you choose *Setup Pens* a window opens which allows you to change pen parameters. The *Check Plot Pens* option allows you to specify a pen number for each of the check plot layers. Note that more than one layer can have the same pen number. You can also specify the number for the main pen and the speed which the pens travel. The figure is given in mils per second and the default speed is 5000. You can also change the size of each of the eight pens.

Printer setup

The procedure for setting up a printer is similar to the procedure for setting up a plotter.

The *Orientation* option has the extra option of *Auto Centre*. This provides for a print to be centered on the printer paper. Note that this option will override any offsets which have been set up.

Another variation is that you can choose between continuous paper and single sheet paper.

You may have to use the *Scale* option if you are using a printer with slightly different characteristics to the Epson FX series.

The Epson FX series has a vertical step size of 1/72 inch. The Epson LQ series, however, has a vertical step size of 1/60 inch. This means that 720 dots is 10 inches long on an FX series printer and 12 inches long on an LQ series printer. The output will therefore be longer in the X direction on an LQ series printer. To make it the same the distance in the X direction must be reduced by a factor of 1.2. Therefore you will need to make the X-correction factor 0.833.

Gerber photoplotter setup

This option allows you to set up the parameters for photoplotter output. The information is sent to a file since it is likely that you will take a disk to a photoplotter bureau to produce the final artwork. Before producing a Gerber file you should read the manual associated with the photoplotter to learn how to set up the required parameters.

GERBER SETUP		
File	:	Gerber
X Offset	:	0
Y Offset	:	0
Match Tolerance	:	25%
Plot Tolerance	:	10%
G54	:	Off

You can supply a file name for the output file by choosing *Setup Gerber File*. The extension for the file is supplied by the program.

A .APT file is supplied which contains the draft codes necessary to produce a plot. The match tolerance gives the percentage variation permissible between the draft codes and the actual pad sizes. The higher the number the greater the difference in area can be.

Some plotters need a 'G54' code to be sent before a draft code is sent. The G54 option allows you to turn this option 'on' and 'off'.

NC drill setup

The process of setting up a file which will drive an NC drill which uses the Excellon format is very similar to that described for Gerber photoplotters in the previous section.

Changing the menu colors

Each of the menu colors can be changed by choosing *Setup Menu Colors*. One nice feature of the program is that you can choose different colors for different levels of menu. Thus you can choose one set of colors for the main menu another set for, say, the *Setup* menu, another

for the *Plotter Setup* menu and another for the *Printer Device* menu.

Notice that after setting up the menu colors for the first level, you can copy the previous setup down to the next level of menu.

Printing or plotting a PCB

The final action you need to take is the actual print or plot. Make sure that your printer or plotter is turned on and is online. Choose *Plot* from the MAIN MENU after making sure that all the *Setup* options are correct and that you have loaded a file. You can check all this in the status window.

The program will check the file and assemble the information relating to the layer or layers to be output. The information window will keep you informed of the progress of the plot.

If the output does not appear within a few minutes, check that the printer is online and plugged into the correct port.

A similar process occurs if you choose Print, Gerber Plot or NC Drill from the MAIN PLOT MENU. Remember that the last two options always produce plot files.

27 THE *PCB3CON* AND *PLIB3CON* PROGRAMS

Program Disk A contains two utilities which allow you to convert files created with *PROTEL-PCB 3.0* so that they are compatible with *PROTEL-AUTOTRAX*.

The *PCB3CON* program

This program will convert PCB files created with *PROTEL-PCB 3.0* so that they can be loaded into *PROTEL-AUTOTRAX*.

The simplest way to convert a file called, say, BOARD.PCB is to return to the DOS prompt and type:

PCB3CON BOARD.PCB NEWBOARD.PCB

Thus, the name of the *PROTEL-AUTOTRAX* file is now NEWBOARD.PCB.

If you do not specify the .PCB extension, that extension will be forced by the program.

If you just type:

PCB3CON and press «ENTER»

then you will be prompted to supply the old file name and the new file name.

The program also asks you whether you want the component designator text and comment text hidden and which pad shape you want.

Remember that the program is stored on Program Disk A, so you have to make sure that this disk and the disk containing the files are present if you have a dual-floppy drive system.

You can specify complete path names for both the old file and the new file.

The *PLIB3CON* program

This program has exactly the same structure as the *PCB3CON* program. It is used to convert libraries that you might have created with *PROTEL-PCB 3.0* so that they are compatible with *PROTEL-AUTOTRAX*.

If you want to convert a library called OLDLIB.LIB to be called NEWLIB.LIB then type:

PLIB3CON OLDLIB.LIB NEWLIB.LIB

All the other features of the program are the same as for *PCB3CON* except that you are only prompted to supply the pad shape.

You need to be aware that libraries created with *PROTEL-PCB 3.0* may increase in size significantly when converted. The reason is that there is more information stored about each pad in *PROTEL-AUTOTRAX* libraries.

APPENDICES

A FEATURES OF *PROTEL-AUTOTRAX*

This appendix outlines the technical features of *PROTEL-AUTOTRAX* which you may need to refer to from time to time.

Important files

The files that are supplied with the package are:

TRAXEDIT.EXE — This is one of the main program files. It must be stored in the same directory as the file TRAXEDIT.OVR.

The file is supplied on Program Disk A.

TRAXEDIT.OVR — This is the 'overlay' part of the program. While this is part of the main program it resides on the disk and is only loaded into memory as required. This helps free up memory space for your boards. This file can be stored in any directory, but it must be in the same directory as the TRAXEDIT.EXE file.

If you have EMS memory installed, the .OVR file is loaded into that portion of memory, thus making calls to the overlay very fast.

The file is supplied on Program Disk B.

TRAXSTD.LIB — This is the main library file which contains the patterns for the components you are likely to place on your boards. Note that SMD devices are included in this library.

This file is supplied on the Library Disk.

TRAXEDIT.PAD	This file contains information about the default pad types. This is the file that is loaded when the program starts. It must be stored in the currently selected directory or a directory pointed to by the DOS Path command. This file is supplied on Program Disk A.
TRAXEDIT.MNU	This file holds the menu messages and the error messages used by *PROTEL-AUTOTRAX* and must always be available to the program at start up. The file is supplied on Program Disk A.
TRAXEDIT.KEY	This file holds the default key macros which are automatically loaded when you start *PROTEL-AUTOTRAX*. The file is supplied on Program Disk A.
DEMO.PCB	This is one of the demonstration PCB files which is referred to in the manual. All PCB files generated with the *PROTEL-AUTOTRAX* program have a .PCB extension. This board has already been routed. This file is supplied on the Library Disk.
PLACED.PCB	This is the unrouted version of DEMO.PCB. This file is supplied on the Library Disk.
DEMO.NET	This is the netlist file with the information which is used to route the PLACED.PCB board. This file is supplied on the Library Disk.
EGA.BAT	A batch file which you can use to ensure that the program uses an EGA graphics driver.
CGA.BAT	A batch file which you can use to ensure that the program uses a CGA graphics driver.

HERC.BAT	A batch file which you can use to ensure that the program uses a Hercules graphics driver.
VEGA.BAT	A batch file which you can use to ensure that the program uses a Vega Deluxe graphics driver.
VGA.BAT	A batch file which you can use to ensure that the program uses a VGA graphics driver.
GRAPH.DRV	The currently selected graphics driver.

The files that you might create have the following extensions:

.PCB	This extension will be attached to any PCB files that you create.
.NET	This extension is attached to the netlist files you create with the ***Highlight Make Netlist*** option in *PROTEL-AUTOTRAX* or those that you have created with the *PROTEL-SCHEMATIC* program.
.PAD	These are the files containing the pad types which you have defined.
.KEY	These are the files you have created which contain the keyboard macros that you have defined.

Files that might be created while you are using the *PROTEL-AUTOTRAX* program are:

???.BAK	This file is created when you save a PCB file. It is a copy of the previous save of the file. If you realise that you have made a major error in the current editing session, you can restore your file to its previous state by renaming the .BAK file.
???.ABK	This is the file which is automatically created every 30 minutes (or whatever you have set the backup time to be in the ***Current Backup*** option). If your system crashes and you need to restore your file from the latest backup, simply rename this file with the name of your current .PCB file.

Hardware and software check list

In order to use the *PROTEL-AUTOTRAX* software you will need the following:

An IBM-PC microcomputer or compatible

2 disk drives (2 floppy drives or one floppy drive and one hard drive) - a hard drive is recommended

Graphics card (CGA, EGA, Hercules, VGA or Vega Deluxe)

Colour monitor

One parallel printer port

Microsoft mouse or compatible (recommended)

Draft printer

MS-DOS or PC-DOS (Version 2.0 or greater)

You will need access to:

A pen and ink plotter

A Gerber-compatible photoplotter

An Excellon format N/C drill

Hardware requirements

To use the *PROTEL-AUTOTRAX* software you will need an IBM-PC/XT/AT or IBM PS/2 microcomputer (or compatible).

The microcomputer should have at least 640K RAM and the software will support expanded memory boards which conform to the L.I.M. standard.

The software will perform adequately on a machine with an 8088 microprocessor running at 4.77 MHz, but you will notice a significant increase in speed if a higher clock speed or a more advanced processor (such as an 80286 or 80386) is used.

Installing a numeric coprocessor will not lead to an increase in speed since the *PROTEL-AUTOTRAX* software only uses integer calculations.

The system should have at least two floppy disk drives or one floppy disk and one hard disk drive. A single drive system is not suitable.

A range of colour cards is supported. The software will work with a Colour Graphics Adaptor (CGA) card, an Enhanced Graphics Adaptor (EGA) card, Hercules card, VGA card and the Vega Deluxe card. Monochrome cards are not supported.

The higher resolution of the EGA, VGA and Vega Deluxe cards make them preferable for intricate design work, however many users have achieved excellent results with CGA systems.

New drivers will be added to this set in the future.

A colour monitor which suits the graphics card is required.

While the software has been designed to work with keyboard-only systems it is strongly recommended that either a serial mouse or a bus mouse be installed. The software works with the Microsoft Mouse or any mouse that is 100% compatible with it.

A dot matrix printer is recommended for draft prints. The software supports the Epson FX and LQ series and the HP Laserjet II.

PROTEL-AUTOTRAX supports the popular HP-GL, DM-PL, and Roland DXY 800 plotter languages. There are numerous plotters available which support these plotting languages, and they span wide price/performance ranges. The following plotters are just a few which have been used successfully by *PROTEL-AUTOTRAX* customers.

HP-7440A (ColourPro), 7475A, 7550A, 7570A, (DraftPro), 7580B, 7585B, 7586B series.

Houston Instruments DMP-40, 41/42, 51/52, 56 series.

Roland DXT 800, DXY 880, 980 series.

Enter Computer Sweet-P Plotters.

Protel PT101 Bench Top Photoplotter

The *PROTEL-AUTOTRAX* software supports Gerber photoplotters and photoplotters that emulate the Gerber series. These photoplotters

produce high quality PCB artwork and are accurate to the *PROTEL-AUTOTRAX* 1 mil resolution.

The software also produces N/C drill files conforming to the Excellon format, including automatic drill changes.

Software requirements

In addition to the *PROTEL-AUTOTRAX* software you will need a copy of either the MS-DOS or PC-DOS operating system. The version number should be 2.0 or greater.

The *PROTEL-AUTOTRAX* software provides automatic backups of your files, but it is strongly recommended that a regular backup of .PCB files is made onto a different disk.

You should use the DOS COPY command to copy your .PCB files to a different disk, then store that disk in a different location. It is also a good idea to back up your .KEY, .LIB and .PAD files from time to time as well.

PROTEL-AUTOTRAX check list

When you receive your *PROTEL-AUTOTRAX* package you should find the following contents:

The *PROTEL-AUTOTRAX* reference manual

Four disks - labelled Program Disk A, Program Disk B, Library Disk and Plot Disk - which contain the *PROTEL-AUTOTRAX* programs and the associated files.

The external protection device, which plugs into the parallel printer port of your microcomputer and allows the *PROTEL-AUTOTRAX* program to work.

The Software License Agreement Form, which you should fill out and return immediately. This is vital if you require further support or information about future updates and revisions.

Parameters

The following list gives the values of many of the *PROTEL-AUTOTRAX* parameters.

Max board size	32 X 32 inches
Board layers	6 signal layers plus power and ground planes
	2 component overlays
	2 solder masks
	2 paste masks
	1 keep out layer
Grid resolution	Visual and snap grid adjustable from 1 mil to 1000 mils
Zoom factor	Variable from 1:1 to 1:100
Track width	Definable from 1 mil to 255 mils
Via size	Definable from 1 mil to 1000 mils
Pad size	Definable from 1 mil to 1000 mils
Pad shapes	Circular, rectangular, rounded-rectangular, octagonal, cross hair and Moire
SMD	Full support for surface mount devices
Nets	The number of nets per file is 1000
Routing grid	Definable from 10 mil to 100 mil
Targets	Moire and cross-hair targets
Cursor shape	Arrow, small cross and large cross
Filename specs	8 characters
Pads	48 pad types

Key macros	48 key macros
Library components	200 components maximum in any one library
Max string size	1000 mils
Min string size	36 mils

B RS232-C CONSIDERATIONS

Many plotters come equipped with a RS232-C interface, and a cable is required between the plotter and computer to provide data transmission and handshaking. Correctly configuring this cable can be an exasperating task - even for experienced engineers. This appendix explains the relevant RS232-C conventions, and gives wiring diagrams for cables that have been tested on a wide variety of plotters and computers with 100% success.

It is natural to ask why parallel interfaces seem to work every time with no problems, whereas serial interfaces can take great amounts of time and patience to set up.

Serial versus parallel interfaces

Parallel interfaces provide one-way transmission only, from a computer to a peripheral device, such as a printer or a plotter. Since data is only transmitted in one direction, the data and handshaking connections and their direction of flow are straight-forward. All data bits (8) originate from the computer and are received by the peripheral. The data strobe signal obviously originates at the computer as well, since it indicates that valid data is on the bus. The acknowledge and busy lines clearly originate from the peripheral, and so on with all of the signals. A standard cable has been defined for all these signals, and the computer and peripheral are intrinsically configured to originate or receive these signals as appropriate.

However, the RS232-C standard defines the signals for bidirectional communication between two pieces of equipment, known as Data Terminal Equipment (DTE). A DTE is defined as:

1. The point of origin for any data to be transmitted, and
2. the final destination for transmitted data.

Thus, the computer and plotter are both defined as DTE. This is where problems begin. From an RS232-C point of view, both the computer and the plotter look the same, and signals such as Transmitted Data (from the DTE) are assigned to the same pins in both devices. Contrast this with the parallel interface in which the interpretation of a signal pin is intrinsic to whether the device originates data or receives it.

Thus, interfacing two DTEs needs an intermediary device or devices to connect the transmitted data pin of one DTE to the received data pin of the other, and vice versa, and to also correctly configure the handshaking signals. The RS232-C protocol defines Data Communications Equipment (DCE), which connects directly to DCE and transmits and receives the data over a channel - a DCE does not originate data, nor is it the final resting place for data. A modem is a DCE that modulates the data for transmission over a voice channel (a telephone line) and demodulates it at the receiving end back to digital data.

Cable connections

In the case of connecting a computer to a nearby peripheral such as a plotter, the cable is configured to take the place of the DCE connecting two DTEs. Because it replaces the modems described previously, such a cable is known as a Null Modem. Figure B-1 (at the end of this chapter) shows the cable configuration required between an IBM-PC/XT (or compatible) and a plotter; Figure B-2 shows the cable required between the IBM-AT (or compatible) and a plotter. The IBM-AT uses a DB9 connector for its serial port, whereas the other machines use the familiar DB25 connector.

Refer to Figure B-1. Note how the cable connects the Transmitted Data (TXD) of one DTE to the Received Data (RXD) of the other and vice versa, to provide the correct flow of data. The handshaking of the plotter is as follows. When the plotter is ready to receive data it asserts the Data Terminal Ready (DTR) signal. This is received at the computer as Clear To Send (CTS) and the computer commences transmission. When the plotter buffer is full, the DTR line goes low, and the computer sees that it is no longer clear to send data. As the plotter continues plotting and empties the buffer, its DTR signal again goes high and the computer recommences data transmission.

From the computer's point of view, it requires that CTS is high as well as Data Set Ready (DSR), Data Carrier Detect (DCD), and (in some computers) Ring Indicator (RI), before it will transmit data. When the computer is turned on, its serial port is initialised and its DTR is asserted, since the computer (DTE) is now ready. By connecting the computer DTR, DSR, DCD, and RI lines within the shell of the connector, the computer can sense that a valid connector is plugged into its serial port.

The cable is then made symmetrical so that either end of the cable can be plugged into the computer or plotter. Users of Houston Instruments DMP plotters will find this handy, since the plotter uses the same male DB25 connector as the IBM-PC/XT. With HP plotters this symmetry isn't as important, since the HP plotters use female DB25 connectors and the cable ends cannot be swapped. Use the correct connectors to plug into your plotter and computer, and follow the connections outlined in Figure B-1.

If you are using the IBM-AT (or compatible) computer, either obtain a DB9 to DB25 adapter and use the cable of Figure B-1, or use the cable of Figure B-2 to connect directly to the DB9 connector on the computer.

This discussion should allow you to connect your computer and plotter without problems, and should enable you to interface with other serial devices as well. Refer to the manual that comes with your plotter for further information on its interfacing and handshaking. The cable given in this appendix may differ slightly from the one given in your plotter manual, but it should be tried initially. It conforms strictly to RS232-C definitions and has been made generally to suit the widest variety of peripherals and compatible computers possible.

Baud rate, data bits, etc.

The other important consideration when using a serial interface is to match the baud rate (110-9600), the number of data bits (7 or 8), stop bits (1 or 2) and the parity bit (Odd, Even, or None) of both the computer and plotter. Your plotter manual should indicate the default settings of the plotter upon being powered up, and will contain information on changing the communications setup. Some plotters do not have default settings but have DIP switch settings for the communication parameters.

Set up your plotter according to the manual and then be sure to match these settings with those in the *Traxplot* program. These settings are stored when the plot finishes, so that once they are set correctly they will not have to be changed while using the same plotter.

For those of you with no idea of how you should initially set your plotter, use the following values to begin with:

Baud Rate:	2400
Data Bits:	8
Stop Bits:	2
Parity:	N

This is an intermediate baud rate and will yield error-free data transmission for reasonably long cables.

What to do in case of problems

1. Don't panic! First check your cable connections, make sure no wires have broken, and check that your *Traxplot* setting matches the baud rate, parity, etc. of your plotter.

2. Make sure you are using the correct serial port. If *Traxplot* specified serial port 1 (COM1:), make sure your plotter is connected to that port and not the parallel port or another serial port.

3. If your plot goes all right for a while and then starts putting stray lines and arcs all over the layout, this generally indicates improper handshaking. Check that your cable is configured exactly as in the appropriate diagram, and check that all communications settings match. This problem can also occur if the data is being corrupted. Keep the plotter cable as short as possible and keep it away from power cords, etc. If you are using a long cable, you may have to reduce the baud rate to obtain error-free transmission. Due to the distributed resistance and capacitance of cables, there is a trade-off between cable length and baud rate for reliable data transmission.

4. Make sure you have specified the correct plotter type in the *Traxplot* program. Is your plotter a true Hewlett-Packard, Houston Instruments, or Roland DXY 800 plotter, or one that emulates one of them? Some plotters, such as the Roland DXY-880, have their own language as well as emulating HP-GL. Make sure that if you are using a dual-language plotter you set the plotter DIP switches for the correct language.

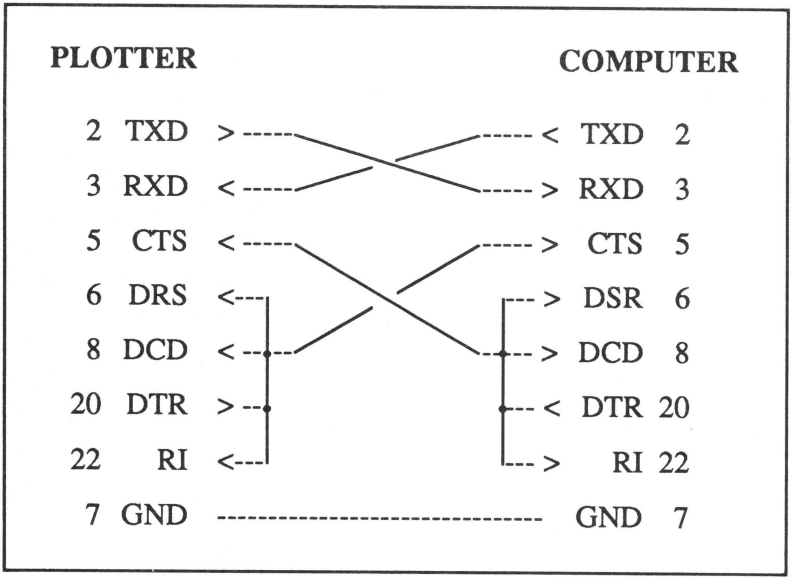


Figure B-1

IBM-PC/XT RS232-C Connection Diagram

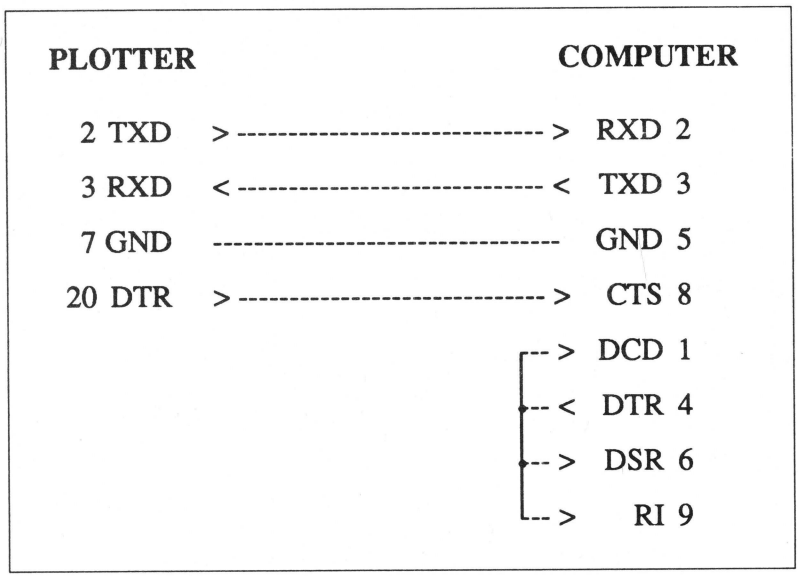


Figure B-2

IBM-AT RS232-C Connection Diagram

C ERROR MESSAGES

This appendix provides a brief explanation of each of the error messages which occur while using the *PROTEL-AUTOTRAX* and *TRAXPLOT* programs. They are arranged in alphabetical order of the first character in the message to make them easy to look up.

Non-fatal errors

BAD FORMAT IN KEY MACRO FILE

The key macro file which has an extension of .KEY has been corrupted. This may have occurred during editing with a word processor.

To overcome the problem, delete the key macro file and copy the backup key macro file into that directory. This remedy assumes that you have made regular backups of your important files.

BAD FORMAT IN PAD TYPE FILE

The pad type file which has an extension of .PAD has been corrupted.

The remedy is the same as for the key macro file.

BAD LIBRARY FILE FORMAT

The library file which has an extension of .LIB has been corrupted.

The remedy is the same as that for the key macro and pad type files.

BAD PCB FILE FORMAT

The PCB file with an extension of .PCB has been corrupted.

The remedy is to delete the file and then copy the latest backup into the directory, or to rename the automatic backup file which is created every 30 minutes under the .ABK extension.

BLOCK NOT MARKED

You have attempted to move, copy or delete a block without first defining a block.

Choose ***Block Define*** to define the block and then carry out the required operation.

COMPONENT DESIGNATOR IS ALREADY USED

You have attempted to assign a component designator that has already been used for another component on the board.

When you place a component the program will automatically supply the next available designator if you type, say, U? instead of, say, U3. If you are placing a number of components of the same type the designator which appears in the input window will automatically be the next available.

COMPONENT NOT FOUND

This error will occur if you attempt a ***Jump Component*** operation to a non-existent component.

Choose ***Information Components*** to check which components are currently placed.

COMPONENT NOT FOUND IN LIBRARY

If you are placing a component or renaming a component and you enter a component name which does not exist in the component library then this error message will appear.

It is possible that you may not have specified the name correctly. To rectify the problem enter a «?» and press «ENTER» or click «LEFT MOUSE». Then select the component you require from the list provided. If the component name does not appear in the list you may need to select another library.

CURRENT LAYER MUST BE TURNED ON

You have attempted to turn a layer 'off' (in *Setup Toggle Layer*) when it is the currently selected layer.

To overcome the problem exit from this menu and select another layer to be the current layer, then try again.

FILE NOT FOUND

You have attempted to load a file which is not in the currently selected directory. This may occur if you do not specify the name of the file correctly.

To overcome the problem, check the name of the file with ***File Files*** or change the path with ***File Path.***

KEY MACRO FILE NOT FOUND

You have specified a key macro file which does not exist.

Choose ***File Files*** and enter the mask *.KEY, then inspect the names of the key macro files on your disk.

KEY MACRO KEY ALREADY USED

You have attempted to assign a key macro to a function key which already has one assigned to it. This may occur in ***Setup Keys Assign*** or ***Setup Keys New.***

If you still want to assign the macro to that key you must first clear the key by choosing ***Setup Keys Delete.***

KEY MACRO NAME ALREADY USED

You have attempted to assign a name to a key macro which is already in use. This may occur in ***Setup Keys Rename*** or ***Setup Keys New.***

To overcome the problem simply enter a different name.

KEY MACRO NOT FOUND

You have specified a key macro which does not exist.

LIBRARY FILE ALREADY EXISTS

This error will occur if you choose *Library·New Library* and try to specify a name for the new library which already exists.

To overcome the problem, change the name you want to give to the new library.

LIBRARY FILE NOT FOUND

You have tried to load a library which has either not yet been created or is not in the directory specified.

LIBRARY FULL

You have attempted to add a component to a library which already has 200 components in it.

To overcome the problem either delete a component from the library first, or create a new library.

LIBRARY NAME IS ALREADY USED

This error occurs during the *Library Add* option if you try to add a new component to a library and assign a name which already exists.

Check the names of the components in the library with *Library List* and then change the name of the new component so that it does not clash with an existing name.

NET NAME NOT FOUND

You might have chosen *Jump Net* and specified a net name which does not exist.

Check that you have supplied the correct net name or that you have loaded the correct netlist using *Netlist Get Nets.*

NETLIST FILE NOT FOUND

You have attempted to load a netlist file in *Netlist Get Nets* which could not be found.

Check the names of the files on the disk. Also check ***File Path*** to make sure that the correct path has been specified. If you are using a floppy disk system make sure that the correct drive has been selected and that you have inserted the correct disk.

NO COMPONENTS IN LIBRARY

You have just created a new library file and then tried to *Library List*, *Library Browse* or *Place Component* or a similar option.

You will need to add some components to the current library or change to a different library.

NO COMPONENTS PLACED

You have attempted to select a component when there are no components placed on the board.

NO KEY MACROS LOADED

An attempt has been made to refer to a key macro when no macros have been loaded. Key macros are stored in a file called PCBEDIT.KEY. This file is automatically loaded when you start the PCBEDIT program.

This error will occur when you try to carry out many of the functions in *Setup Keys* if you have cleared the key macros from the memory.

To overcome the problem, choose *Setup Keys Load* and load in a key macro file.

NO LIBRARY HAS BEEN OPENED

You have tried to load a component or save a component without first having specified a library.

Normally, the default library is opened when you start *PROTEL-AUTOTRAX*. If no library is open check your disk or directory to make sure a library file exists.

NO NETS LOADED

You have attempted to carry out a netlist operation such as *Netlist Optimize* or *Netlist List* when there are no nets currently in the memory.

You will need to load some nets from a .NET file by using *Netlist Get Nets*.

NO OUTLINE ON KEEPOUT LAYER

You have attempted to place components automatically using *Netlist Auto Place* without having specified the outline of the board on the keep out layer.

Use *Place Track* to put an outline on the keep out layer.

NO PAD TYPES LOADED

The .PAD file should be loaded automatically when you start *PROTEL-AUTOTRAX*.

Make sure the .PAD file is on the correct floppy disk or is in the correct directory.

NO PINS HAVE BEEN HIGHLIGHTED

You have chosen the *Information Highlighted Pins* option without having first highlighted any pins.

To rectify the problem choose *Highlight Net* or *Highlight Connection* and highlight some pins.

NO PINS ON THAT PLANE

You have tried to list the nets on the power or ground plane when no connections have been made.

Check that you really want the connections to be on that plane. If you do, then turn that plane on with *Setup Toggle Layers* and make the required connections.

NO PRIMITIVES FOR LIBRARY COMPONENT

You have tried to save a component to a library but have not yet defined any primitives such as pads or tracks for that component.

NOT ENOUGH MEMORY FOR COMPONENT

You have tried to create a component or merge a component when there is not enough memory available.

If you need the new component then delete one of the existing components before loading the new one.

NOT ENOUGH MEMORY TO SUSPEND TO DOS

There is not enough memory left to exit to Dos.

You should either choose ***File Quit*** or continue working without exiting to Dos.

ONLY 48 KEY MACROS AT ANY TIME

An attempt has been made to define a 49th key macro. *PROTEL-AUTOTRAX* only allows 48 key macros to be defined at any time.

To overcome the problem choose ***Setup Keys Delete*** and delete a macro you no longer need before defining the new macro.

ONLY 48 PAD TYPES AT ANY TIME

You have attempted to define a new pad type and 48 have already been defined. A maximum of 48 pad types is allowed in the *PROTEL-AUTOTRAX* program.

If you decide that you need the new pad type you must first delete one of the existing types or change one of the existing types using the ***Edit Pad*** option.

OUT OF MEMORY

There is not enough memory for the operation you have attempted.

This is likely to happen if you attempt to load a large .PCB file.

PAD NOT FOUND

You have chosen the ***Jump Pad*** option and specified a pad number which does not exist on the chosen component.

An example would be selecting a DIP16 pattern and then specifying pin 17 for the jump.

Check the number of pins in the component and specify a pin which exists.

PAD TYPE FILE NOT FOUND

The pad type definitions are stored in a file with a .PAD extension. This file must be in the same directory as the .EXE file.

Check that the file has not been deleted or moved to a different directory.

PAD TYPE NAME ALREADY USED

You have attempted to give a name to a new pad when that name is already in use.

Either change the name of the new pad or use ***Edit Pad*** to change the name of the existing pad.

PAD TYPE NOT FOUND

You have tried to select a pad type which does not exist.

PATH NOT FOUND

You have attempted to specify a non-existent drive or directory in the ***File Path*** option.

Check the path specification to ensure that both the drive and the directory actually exist.

PCB FILE NOT FOUND

An attempt has been made in ***File Load*** to load a PCB file which does not exist or which cannot be found.

Check the names of the PCB files with the ***File Files*** option. If the filename is not displayed, then check ***File Path*** to see if the correct pathname has been specified. If you are using a floppy disk system, check to see that the correct drive is specified and that the right disk has been inserted.

SMD PADS ON OUTSIDE LAYERS ONLY

You have tried to place a Surface Mount Device (SMD) on an inside layer of the board.

An SMD can only be assigned to either the top layer or the bottom layer and belongs to only one of those layers.

STRING NOT FOUND

You have specified a string in the *Jump String* option which is not on the board.

This may be because the string you specified is not present or because you have specified it incorrectly or incompletely.

Fatal errors

In each case, these errors cause the program to stop operating. You will need to start the program again and retrieve the latest version of your .PCB file from the backup version.

If you generate a fatal error and cannot work out the reason for it, we would appreciate it if you contacted Protel Technology Pty Ltd.

Division By Zero Error. TRAXEDIT Program Aborted.

The program has attempted to carry out a fatal division by zero operation. Dos traps such an error and causes the program to stop operating.

Stack Overflow Error. TRAXEDIT Program Aborted.

The stack has overflowed for some reason. This is a rare error.

Not Enough Memory. TRAXEDIT Program Aborted.

You have run out of memory, possibly because your PCB file has become too large.

Floating Point Overflow Error. TRAXEDIT Program Aborted.

Another rare error over which you have no control.

Floating Point Underflow Error. TRAXEDIT Program Aborted.

Same as for Overflow error

Not Enough Memory To Start TRAXEDIT.

This suggests that one or more resident programs are already in the memory. You are likely to generate this error if you suspend to Dos by choosing ***File Dos*** and then try to start the program by typing TRAXEDIT instead of EXIT.

TRAXEDIT.MNU Not Found. TRAXEDIT Program Aborted.

All of the menu descriptions and messages are held in a file called TRAXEDIT.MNU. This file must be in the currently selected directory when the program is started.

Security Device Not Found, Program Aborted

You have attempted to run the *PROTEL-AUTOTRAX* program without having first connected the external security device supplied with the program.

D A NOTE TO EDUCATORS

If you are introducing computer-based printed circuit board design to students then there are a number of factors which you might want to take into consideration.

With packages of the sophistication of *PROTEL-AUTOTRAX* within the price range of most PCB designers, there seems little point in spending too long teaching students about the traditional methods of designing boards.

There are, however, some very good reasons why students should be familiar with the basic concepts of printed circuit boards before they begin designing them on the computer.

Concepts such as layers, tracks, pads, vias and components should be thoroughly understood before the students attempt to design their own board using the *PROTEL-AUTOTRAX* software.

These concepts can best be understood by being involved in the production of a simple printed circuit board. Students need to be able to see a board and be able to understand how the components are connected and why the tracks are laid the way they are. They need to see the pads and vias and how they relate to the tracks and components.

Before the students design their own boards, they need to become familiar with the software. It is assumed that they have already learned how to use the keyboard and are familiar at an elementary level with the basic DOS commands. They can learn much by using the demonstration boards and netlist supplied with the package.

The first set of skills to acquire could be:

starting the *PROTEL-AUTOTRAX* program;

loading the DEMO file;

using the drop-down menus;

selecting with the selection bar and with keystrokes;

using the cursor keys or mouse to move around the board;

speeding up cursor movement with the «SHIFT» key;

seeking information about the board from the ***Current***, ***Information***, ***Edit*** and ***Setup*** menus;

locating components, tracks, vias and pads;

changing some of the ***Setup*** options;

inspecting the library components with ***Library Browse***;

saving the current board;

quitting from the program.

Notice that all of these skills can be acquired before any design work is carried out. The students can also become familiar with the *TRAXPLOT* program at this stage. They can learn the following skills:

loading the *TRAXPLOT* program;

loading the file to be output;

choosing the correct port for the printer;

choosing the correct printer;

choosing the setup options for the output;

producing the draft output.

Once these basic skills have been acquired, the students can then move to the computer-based design stage. Tutorial two and tutorial three contain much of the basic information required for the design stage.

It is important to introduce the students to the concepts of netlists and automatic placement and automatic routing early as these features of *PROTEL-AUTOTRAX* are not only very powerful, but also make the whole design process easier for students to understand.

INDEX

INDEX

* key 31, 38
+ key 32
- key 32

Absolute cursor mode 69
Absolute origin 69-70
Adding a component to the library 48, 103
Alt key 22, 142
Any node routes 123
Arcs, deleting 73-4
Arcs, moving 109-10
Arcs, placing 47, 131-2
Arrow keys 22
Assigning comments 35, 133, 140
Assigning component designators 35, 133, 140
Assigning key macros 142-3
Auto pan option 146, 155
Auto Place 78-9, 116
Automatic backups 14, 146, 177
Automatic component placement 78-9, 116
Automatic via placement 129, 148

Backing up files 14, 146, 177
Backspace key 22
Block menu options 63-8
Blocks, copying 67
Blocks, defining 63-4
Blocks, hiding 64
Blocks, moving 64-8
Blocks, reading 68
Blocks, saving 68
Board dimensions 95-6
Board status 97-8
Bottom layer 151
Bottom overlay 152
Breaking a track 110
Browsing in the library 103-4

C routes 123
CGA graphics driver 16, 141
Changing 'default' setup options 139-52
Changing layers 31-2, 38, 70
Changing the menu window colour 30-1, 141
Changing the current pad type 70-1
Changing the cursor type 147
Changing the drag option 147
Changing the grid size 89-90
Changing the layer colors 140-1
Changing name of the automatic

backup file 146
Changing the redraw quality 150
Changing the time between automatic backups 146
Changing track widths 71, 80, 125
Checking the pins in a net 96
Checking the pins on a component 37,101
Chevrons « » 21
Choosing a component from the library 34-5, 103
Choosing a different library 106-7
Choosing options from menus 24, 32
Clearing a file 34, 83
Clearing a netlist 117
Clearing key macros 53, 143
Clearing the work space 34, 83
Colors, layers 140-1
Colors, menu window 141
Comments 35, 133, 140
Compacting the library 105
Component designator 35, 132-3, 140
Component designator, incrementing 67, 132-3
Component text layer 139-40
Components, 'seed' 116
Components, adding to the library 48, 103
Components, comments 35, 133, 140
Components, deleting 74
Components, editing 77-9
Components, free to move 78
Components, jumping to 99-100
Components, listing in library 96-7
Components, listing those placed on the board 96
Components, locked 78-9, 116
Components, moving 110
Components, placing 132-3
Components, placing automatically 78-9, 116
Connections, highlighting 91-2
Connections, length 119
Contracting the display 31, 156
Converting Protel-PCB 3.0 files 171
Converting Protel-PCB library 3.0 files 172
Copying a block 67
Creating a new .PCB file 34
Creating a new library 107
Creating a new pad type 44-5, 150
Creating key macros 53-6, 141-5
Cross-hair target 135
Ctrl key 22, 142
Current menu option 69-72
Current pad type 70-1
Cursor keys 22
Cursor mode absolute 69
Cursor mode relative 69
Cursor types 147

Cycling through the layers 31-2, 38, 70

Defining a block 63-4
Defining macros 54-5, 141-5
Defining pads 45
Del key 22
Delete menu option 73-6
Deleting a block 67
Deleting a library component 105-6
Deleting arcs 73-4
Deleting components 74
Deleting fills 74
Deleting highlights 75
Deleting key macros 143
Deleting pad types 148
Deleting pads 75
Deleting strings 75
Deleting tracks 76
Deleting vias 76
Design rule checking 117, 124
Displaying the whole board 156
DOS exit 84, 162
Drag option 147
Dragging tracks 111, 112
Duplicating a net 92-3

Edge connectors 116
Edit menu option 77-82
Editing a component 27-8, 77-9
Editing a pad 79-80
Editing a pad type 148-9
Editing a string 80-1
Editing a track 80
Editing a via 81-2
Editing key macros 143
EGA graphics driver 16
END key 31, 37, 155
Enter key 22
Error messages 189-98
Excellon drill 169
Exiting from *PROTEL-AUTOTRAX* 24, 87
Expanded memory board 14
Expanding the display 25-6, 31, 156
Exploding a component 106
External plane 133-4
External protection device 14, 180

Features of *PROTEL-AUTOTRAX* 175-82
File menu option 83-8
File types 175-7
Files, listing 84-5
Files, loading 23, 85-6
Filling an area with copper 134-5
Fills, deleting 74
Fills, moving 112
Finish recording a macro 145

Floating origin 69
Freeing a component 78
Function keys 22, 53-6, 141-5

Gerber photoplotter 119
Graphics drivers, setting up 16, 17
Grid menu option 89-90
Grid, snap 44, 89-90
Grid, visible 90
Ground net 97, 120
Ground plane 41, 69-80, 97, 152

Hardware requirements 13, 178-80
Hiding blocks 64
Hiding component text 139-40
Hiding nets 118
Highlight menu option 95-8
Highlight, duplicating 92-3
Highlight, make netlist 93-4
Highlighting a connection 91-2
Highlighting nets 93
Highlights, deleting 75
HOME key 31
Horizontal routes 123

Identifying nets 119
Information menu option 95-8
Information window 26
Input window 23, 29, 50
Installing *PROTEL-AUTOTRAX* on a hard disk 15-6
Invisible grid 44

Jump menu option 99-102
Jumping to a component 29-30, 99-100
Jumping to a location 100
Jumping to a net 100
Jumping to a pad 30, 37, 101
Jumping to a pin 30, 37, 101
Jumping to a string 101-2
Jumping to the origin 101

Keep out layer 116, 126, 152
Key macros 53-6, 141-5
Key macros, clearing 143
Key macros, creating new 144-5
Key macros, deleting 143
Key macros, editing 143
Key macros, listing 144
Key macros, loading 144
Key macros, recording 145
Key macros, saving 145

L routes 123
Layers, on and off 151-2
Layers, choosing colors 140-1
Layers, component text 139-40
Layers, description 151

Layers, switching between 31-2, 38, 70
Laying tracks 37-8, 45, 135-6
Leaving *PROTEL-AUTOTRAX* 87
Left mouse button 22, 24
Length of a connection 119
Libraries, merging 107
Library menu option 103-8
Library, adding a component 48, 103
Library, browsing 103-4
Library, creating a new library 107
Library, deleting a component 105-6
Library, exploding a component 106
Library, listing the components 105-6
Library, loading a library file 106-7
Library, merge 107
Library, renaming a component 108
Line editor 49-52
Listing components in the library 107
Listing components on the board 96
Listing key macros 144
Listing PCB filenames 84-5
Listing the current pad types 70-1, 149
Loading a library component 132-3
Loading a library file 106
Loading a netlist 40, 117-8
Loading a PCB file 23, 85-6
Loading a PCB file to print or plot 58, 163-4
Loading key macros 144
Loading pad types 149-50
Locking a component 78-9

Macros (see key macros)
Main menu 24
Making a netlist 93
Memory capacity 14
Memory routes 123
Merging libraries 107
Merging pad types 149-50
Mid layer 151
Minimising number of vias 124, 128
Moire target 135
Mouse buttons 22, 24
Move menu option 109-114
Moving a block 64-7
Moving arcs 109
Moving around the board 155-8
Moving components 110
Moving fills 112
Moving pads 112
Moving strings 113
Moving tracks 110-12, 113
Moving vias 114
Multi layer 152

Nested macros 56
Netlist 38, 115

Netlists, auto place 116
Netlists, clearing 117
Netlist menu option 115-30
Netlists, connection length 119
Netlists, creating 39, 93-4, 120
Netlists, loading 40, 117-8
Netlists, optimizing 40, 118
Nets, displaying 40
Nets, duplicating 93
Nets, hiding 118
Nets, highlighting 93, 119-20
Nets, identifying 119
Nets, jumping 100-1
Nets, listing 97
Numeric keypad 22

Optimizing, X bias 120
Optimizing, Y bias 120
Optimizing netlists 40, 119-20
Origin absolute 69-70
Origin floating 69-70
Orthogonal mode - turning it off 136, 147
Orthogonal mode 136, 147

Pad to pad routing 39, 129-30
Pads, changing the current type 70-1
Pads, clearing currently defined types 148
Pads, defining new shapes 44-5, 150
Pads, definitions 148-50
Pads, deleting 75
Pads, deleting pad types 148
Pads, editing 79
Pads, editing a pad type 148-9
Pads, listing current types 70-1, 149
Pads, loading pad types from a file 149-50
Pads, merging pad types from a file 149-50
Pads, moving 112
Pads, orientation 71
Pads, placing 45-6, 135
Pads, rotating 71
Pads, saving pad types 150
Pan 155
Path command 86, 163
PGDN key 31, 36, 156
PGUP key 31, 156
Place menu option 131-6
Placing a fill 134-5
Placing a string 135, 151, 152
Placing an external plane 133-4
Placing arcs 47, 131-2
Placing components 34-5, 132-3
Placing components automatically 78-9, 116
Placing pads 45-6, 135
Placing text 135, 151, 152
Placing tracks 37-8, 45, 135-6

Placing vias 136
Plotting 57-60
Power net 97, 120
Power plane 42, 97-80, 152
Pre-routes 122
Printing 57-6
Prompt line 27, 126
PROTEL-AUTOTRAX files 175-7
PROTEL-AUTOTRAX parameters 181-2

Question mark on input line 29
Quitting from *PROTEL-AUTOTRAX* 24-5, 87
Quitting from *TRAXPLOT 163-4*
Quitting to DOS 84, 162

Ratsnest 40, 120, 126
Re-centering the screen over the cursor 31
Re-routing a track 113
Reading a block 68
Recording a key macro 55-6, 145
Recursion depth counter 94, 126, 134
Redraw quality 150
Redrawing the screen 31, 155
Reference point 64
Relative cursor mode 69
Renaming a library component 108
Renaming the backup file 147
Repeat menu option 137-8
Repeating placements 137-8
Restoring deleted items 153-4
Returning from DOS 84
Right mouse button 22
Rotating a string 66
Rotating a block 66
Rotating a component 36, 66
Rotating pads 71
Route, manual 128, 129
Route, net 129
Route, one connection 128
Route, pad to pad 39, 129-30
Route, passes 122-4
Route, separation setup 124-5
Route, setting up layers for 121-2
Route, setting up variables 125
Routing a board 39-42, 115, 121-30
RS232-C 183-8

Saving a block 68
Saving a board 38, 87-8
Saving a PCB file 38, 87-8
Saving key macros 145
Saving pad types 150
Seed components 116
Selecting a component from the library 34-5, 132-3
Selecting a port for plotting or printing 167

Selecting a plotter 167
Selecting a printer 167-8
Selecting current layer 32, 70
Selecting current pad 70-1
Selecting layers 31-2, 151-2
Selecting options from menus 24, 32
Selection bar 24
Serial communication 183-8
Setting options 146-8
Setting string defaults 150
Setting the cursor mode 69
Setting up component text 139-40
Setting up layers for routing 121-2
Setting up router passes 122-4
Setting up the plotting options 165-6
Setup menu option 139-52
Shift key 22, 142
Showing nets 120
Single key menu selection 32
SMD (See surface mount device)
Snapping to the snap grid 44
Software License Agreement 15, 180
Software requirements 14, 178, 180
Starting *PROTEL-AUTOTRAX* 17, 23
Status line 24, 89, 126, 134, 157
Status of board 26-7, 97-8
Status window, *TRAXPLOT* 161, 163
Stop recording a macro (CTRL END) 56, 145
Strings, deleting 75
Strings, editing 80-1
Strings, jumping 102
Strings, moving 113
Strings, placing 135, 151, 152
Strings, rotating 66
Surface Mount Devices 78, 149, 151
Switching layers 31-2, 38

Text height 72, 81
Text width 72, 81
Top layer 151
Top overlay 151
Track clearance 125
Track width 71, 80, 125
Tracks, breaking 110
Tracks, deleting 76
Tracks, editing 80
Tracks, moving 110-112, 113
Tracks, placing 135-6
Tracks, re-routing 113
Turning layers on and off 151-2

Un-delete menu option 153-4
Using the keyboard 21-2

Variables, routing 125
VEGA deluxe graphics driver 16
Vertical routes 123

VGA graphics driver 16
Via hole size 72, 81-2
Via minimisation 124, 128
Via size 72, 81-2, 125
Vias, deleting 76
Vias, editing 81-2
Vias, moving 114
Vias, placing 136
Viewing library components 103-4

Z routes 123
Zoom menu option 155-8
Zoom value 157
Zoom window 157-8
Zooming 25, 55, 155-8